HEIDENHAIN

TNC 640

Contouring Control for
Machining Centers and
Milling/Turning Machines

HSCI

March 2014

This brochure describes the functions and specifications of the TNC 640 with NC software 34059x-04

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.

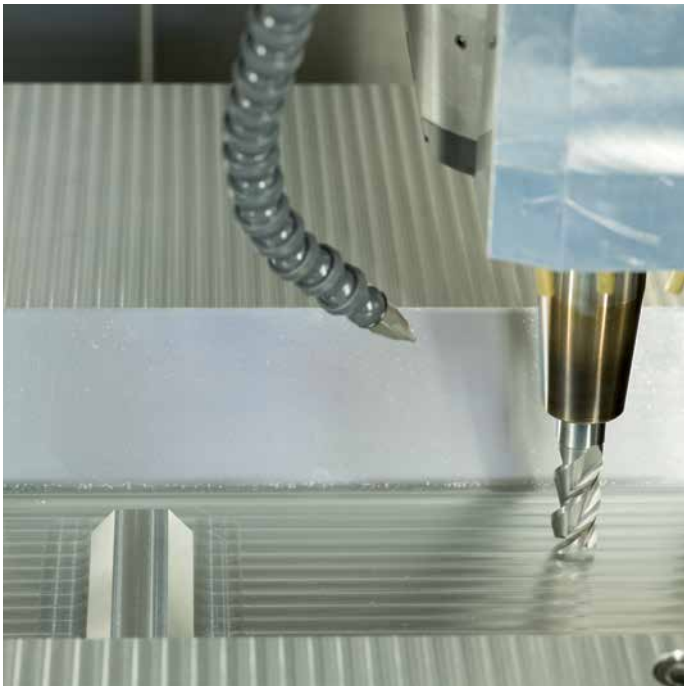
You will find these principles also implemented in the TNC 640, the HEIDENHAIN contouring control for milling and milling/turning operations: shop-oriented programmability with graphic support, many field-proven cycles and an operational design you'll recognize from other HEIDENHAIN controls.

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.

Easy to operate
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 Ethernet interface guarantees very short transfer times, even of long programs.



Universally applicable
The TNC 640 is particularly well suited for milling, turning, HSC and 5-axis machining 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 with smoothed 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
- Cylinder 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
- Datum management
- 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

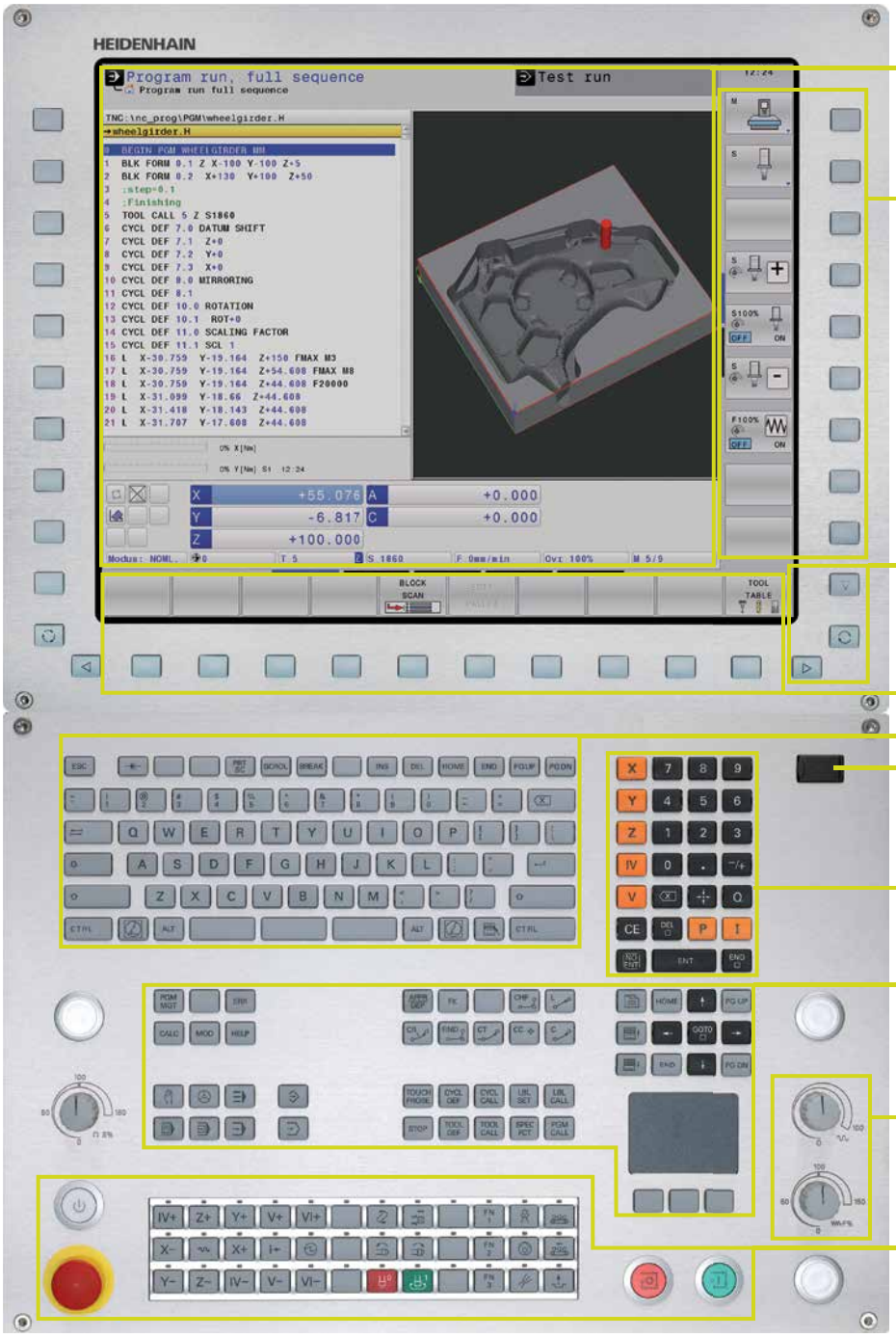
The screen
The large 19-inch TFT color flat panel display with shows a clear overview of all relevant information for programming, operating and inspecting the machine tool and control: Program blocks, notes, error messages, etc. More information is provided through graphic support during program entry, 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 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 **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), 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**

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

Override potentiometers for feed rate, rapid traverse and spindle speed

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 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.

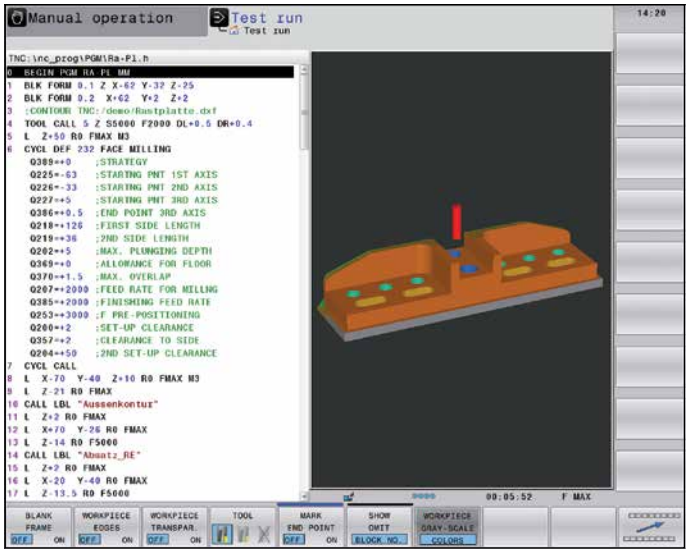
Well designed and user friendly
– The functional user interface

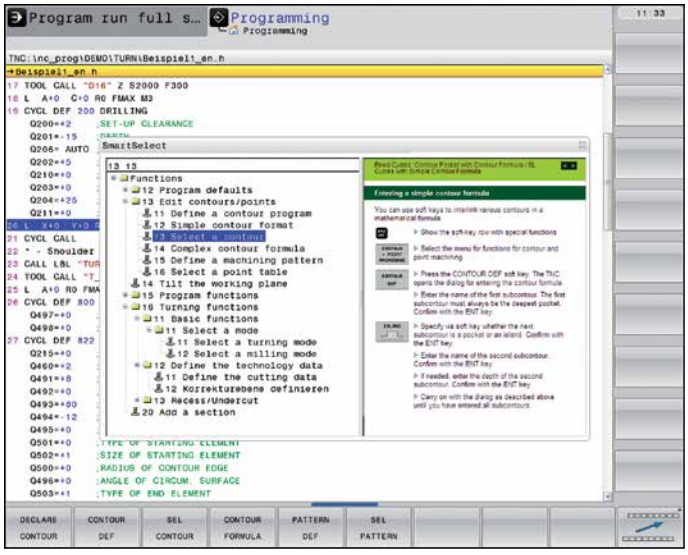
The combination of the straightforward and ergonomically designed keyboard and the well-designed screen layout are the essence of reliable and fatigue-free operation. These are principles that HEIDENHAIN controls have always embodied. However, the TNC 640 also offers a number of features that make working 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 distinguished and the operating modes are also indicated by their respective symbols.

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.



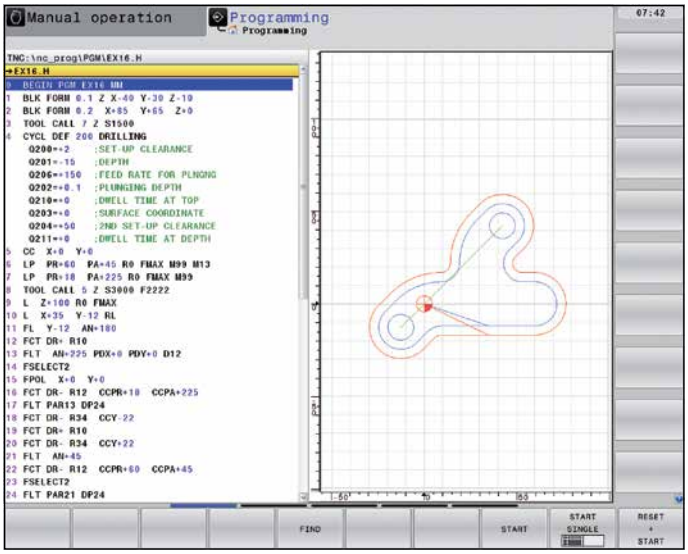


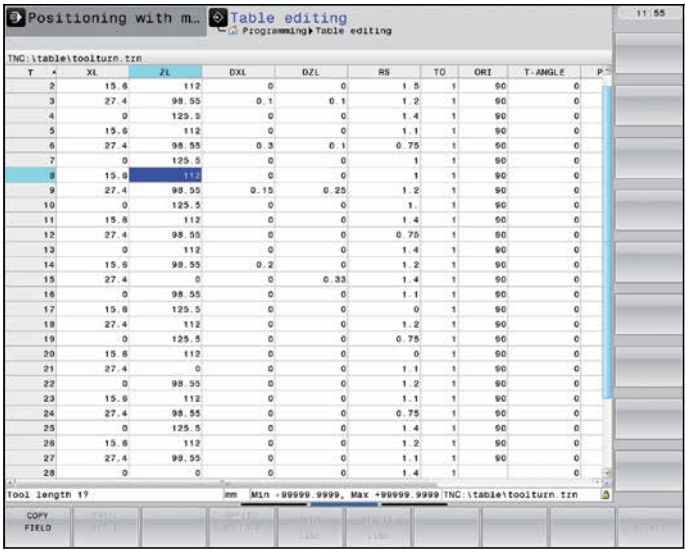
Color-structured programs
The content of a program line can be quite comprehensive: line number, program function, input values, comment. To help you always find your way even in complex programs, the individual program elements on the TNC 640 are shown in different colors. The color syntax highlighting 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, datum 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 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? Here the TNC 640 helps you to save a great deal of time: On a milling turning machine with the TNC 640, you machine the workpiece from a single casting: 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.

If a contour exists in DXF format, you can easily import it with the aid of the DXF converters (option).

***** 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, thread turning and recess turning. The field-proven HEIDENHAIN lathe controls provide 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 accuracy
– 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 accuracy 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. And the TNC 640 interpolates simultaneously in up to five axes. To attain the required cutting speeds, the TNC 640 control spindle speeds up to **60 000 min^{-1}** digitally.

Highest contour accuracy and surface quality
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.

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

The Advanced Dynamic Prediction feature (ADP) expands the conventional look-ahead of the permissible maximum feed rate profile and makes optimized motion control possible to produce clean surfaces and perfect contours. ADP shows its strengths for example during bidirectional finish milling through symmetrical feed behavior on the forward and reverse paths as well as through particularly smooth feed rate curves on parallel milling paths.

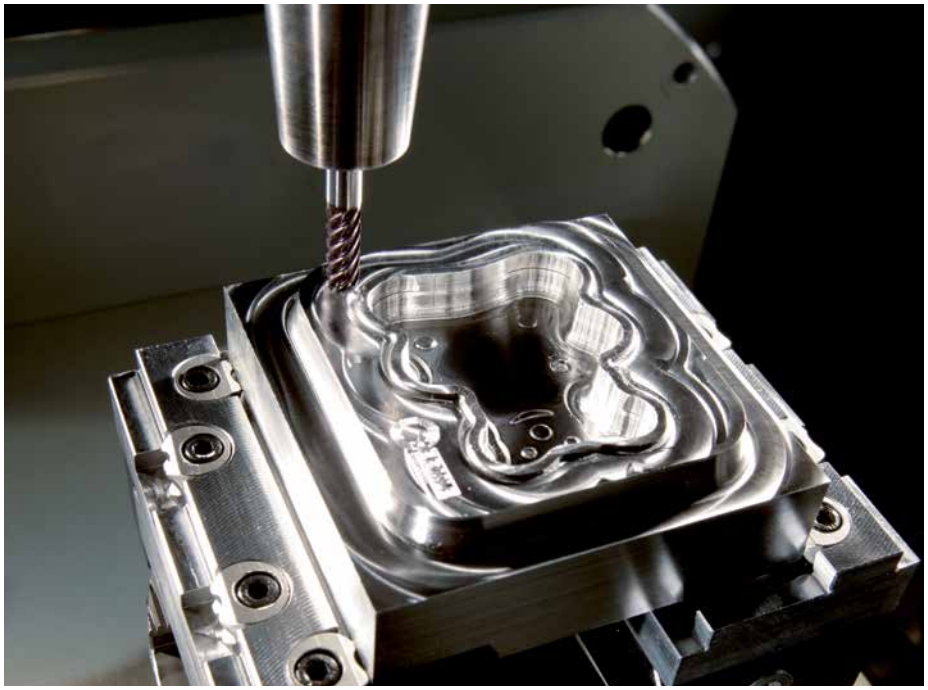
NC programs that are generated on CAM systems (negatively) influence the machining process through various factors such as short step-like contours, coarse chord tolerances and heavily rounded end-point coordinates. Through an improved reaction to such influence quantities and the exact fulfillment of dynamic machine parameters, ADP not only improves the surface quality of the workpiece, it also optimizes the machining time.

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 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 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 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.



Quick and reliable machining with high contour accuracy
– 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 the 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 through compliance between the machine and TCP, thereby increasing 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

Machining with five axes
– The TNC 640 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 path control, its precalculation of the contour and its algorithms for jerk limitation, the TNC 640 has the right functions for a perfect surface in the shortest possible machining time. See for yourself. In the end, it's the quality of the workpiece that proves the performance of the control.

3-D contour machining at its finest
The TNC 640'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 TNC 640 looks ahead and thinks with you. Its "look-ahead" function anticipates future changes in direction by adjusting the traversing speed to the programmed surface. It also enables the TNC 640—if desired—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 TNC 640 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 TNC 640 automatically calculates a 3-D tool compensation (optional) for end mills, ball-nose cutters, or toroid cutters.

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. When machining with five axes (three linear axes and two tilting axes*), the tool can stay perpendicular, or if desired, inclined at a predetermined angle to the workpiece surface.

Regardless of what type of 5-axis programs you wish to run, the TNC 640 makes all the compensating movements in the linear axes that result from movements in the tilting axes. The TNC 640's **Tool Center Point Management** feature (TCPM)—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 TNC by the machine tool builder.

With TCPM you can define the behavior of the tilting and compensating movements automatically calculated by the TNC 640.

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**, machining is mainly by the side of the tool. The tool tip also travels on a straight path, but additionally the tool's circumference machines an explicitly defined plane.

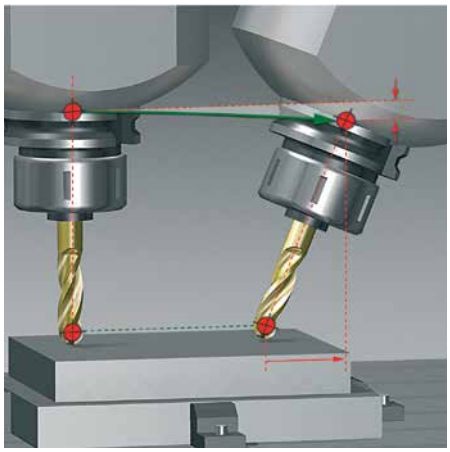
TCPM defines the **effect of the programmed feed rate** as desired either

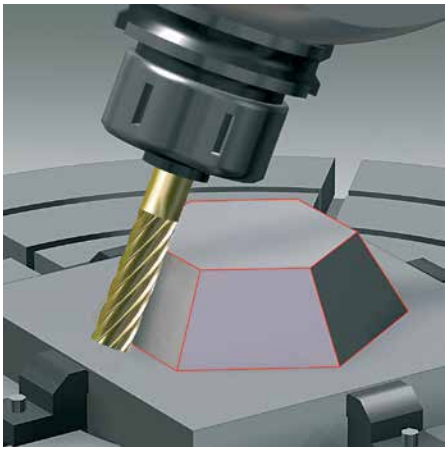
- as the actual velocity of the tool tip relative to the workpiece. Very high axis feed rates can result from large compensating motions during machining near the center of tilting.
- as contouring feed rate of the axes programmed in the NC block. The feed rate is usually lower, but you attain better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

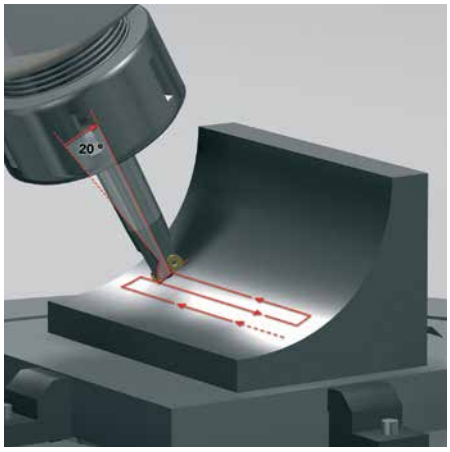
With TCPM you can also define the **effect of the inclination angle** for more uniform cutting passes when working with an inclined radius cutter:

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

The TNC 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 TNC 640 makes sure that the tool remains on the contour and does not damage the workpiece.







Machining with five axes
– Swivel head and rotary table controlled by the TNC

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 TNC 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 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.

With the PLANE feature, defining a tilted working plane is easy: 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 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 TNC by the machine tool builder.

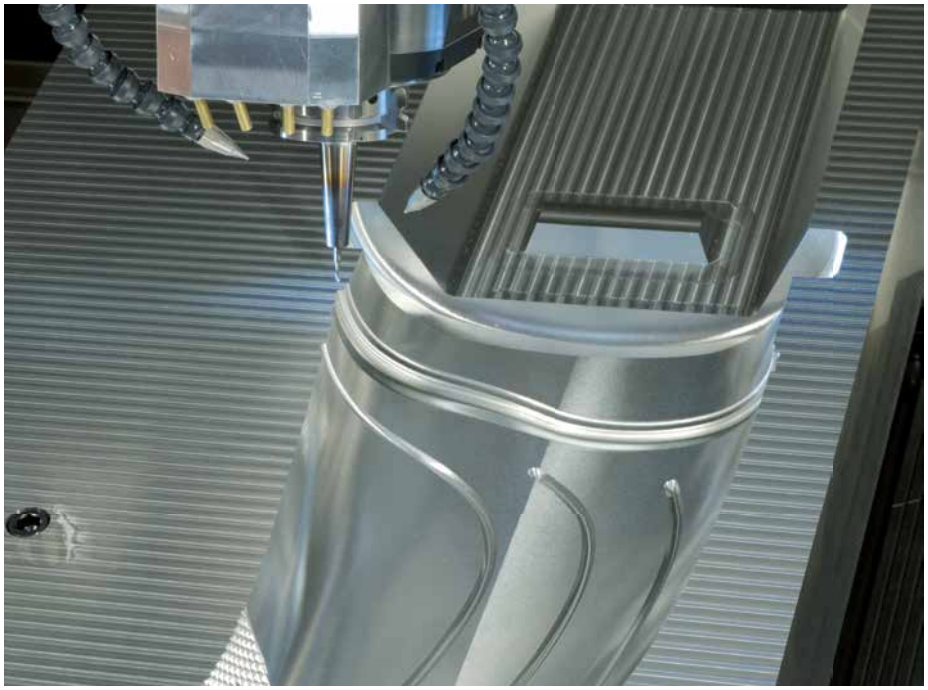
Machining cylindrical surfaces*
With the TNC 640 it is quite easy to program contours (consisting of straight lines and arcs) on cylindrical surfaces using rotary and tilting tables: You simply program the contour in a plane as if the cylinder surface were unrolled. You enter a contour in two dimensions—as if in a plane—and the TNC 640 then calculates and machines the corresponding cylindrical contour.

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

Manual axis motion in the tool direction on 5-axis machines
The safe retraction of a tool is very important with 5-axis machining. The "Virtual Tool Axis" function is of assistance here. You can use it to traverse the tool in the current direction of the tool axis 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.

Linear feed rate for rotary tables in mm/min*
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.

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



Intelligent machining
– Dynamic Collision Monitoring option (DCM)

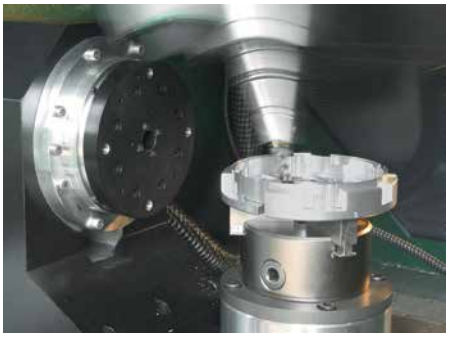
The complex motions and high traversing speeds of five-axis machining make axis movements difficult to foresee. This makes collision monitoring a valuable function that relieves the machine operator and protects the machine from damage.

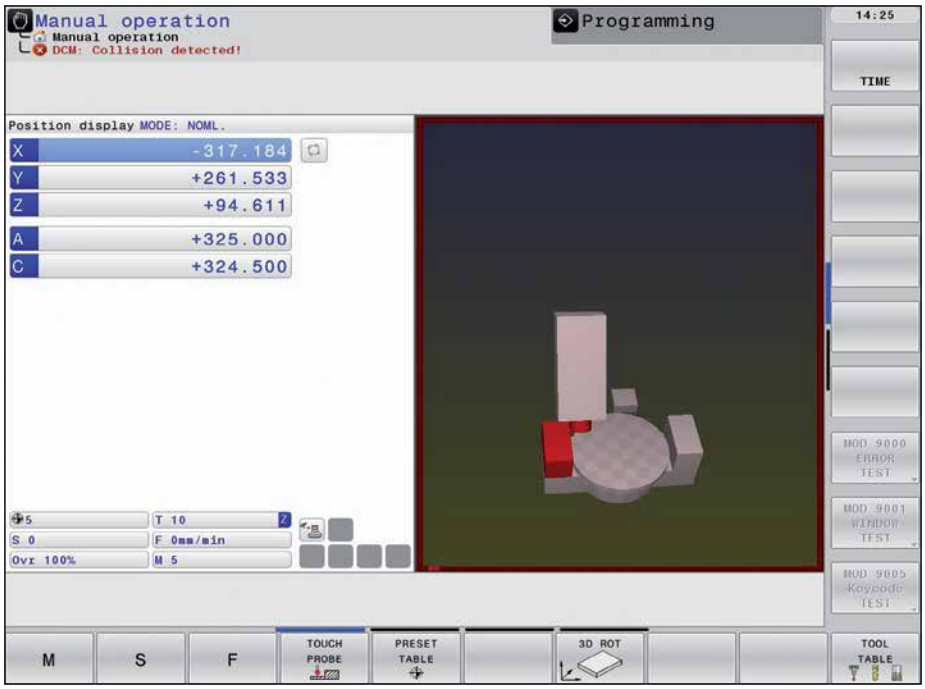
NC programs from CAM systems may avoid collisions of the tool or tool holder with the workpiece, but unless you invest in expensive offline machine simulation software, they ignore the machine components located within the work envelope. And even then it cannot be guaranteed that machine conditions,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

In cases such as these, the machine operator is supported by the **dynamic collision monitoring (DCM)*** feature of the TNC 640. The control interrupts machining whenever a collision is imminent, thereby increasing safety for the machine and its operator. This helps to prevent machine damage, which can result in costly downtimes. Unattended shifts become safer and more reliable.

However, DCM works not only in **automatic mode**. It is also active in **manual operation**. If, for example, during setup the machine operator takes a collision course, the TNC 640 detects it, stops axis movement, and issues an error message.

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





Of course the TNC 640 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.

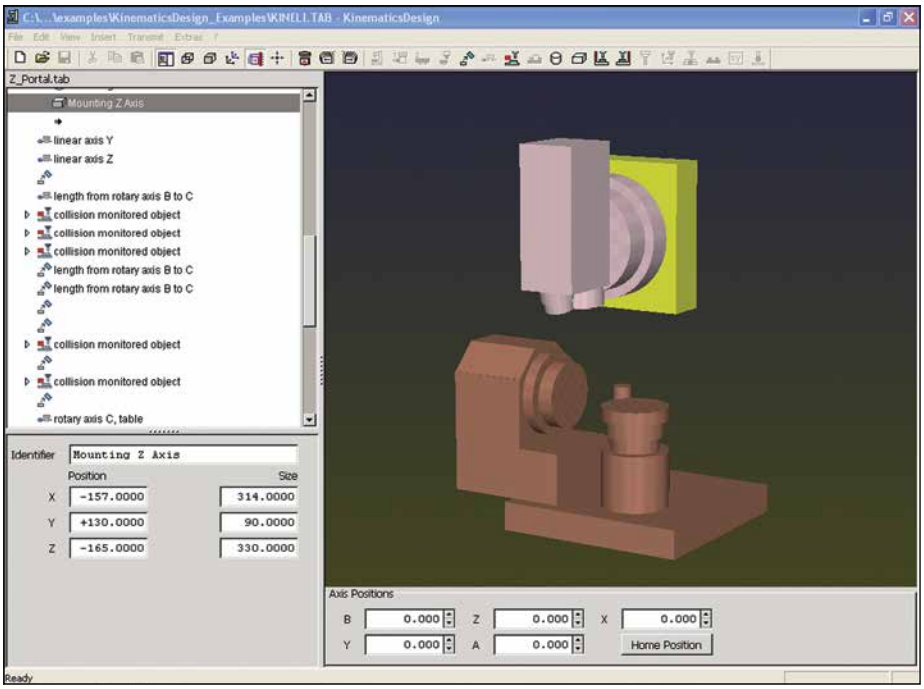
The machine tool builder takes care of the required definition of **machine components.** The working space and the collision objects are described using geometric bodies such as planes, cubes and cylinders. Complex machine components can be modeled with multiple geometric bodies.

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 description of 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 collide with the machine cabin.

When using the dynamic collision monitoring, please note:

- DCM can help to reduce the danger of collision. However, DCM cannot completely prevent collisions.
- Only the machine manufacturer can define machine components.
- Collisions between machine components (such as swivel heads) and the workpiece cannot be detected.
- DCM cannot be used during operation in following error mode (which means without feedforward).



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 TNC functions:

- **Active Chatter Control** (ACC). This option reduces chatter tendencies and permits greater infeeds
- **Adaptive Feed Control** (AFC). This 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

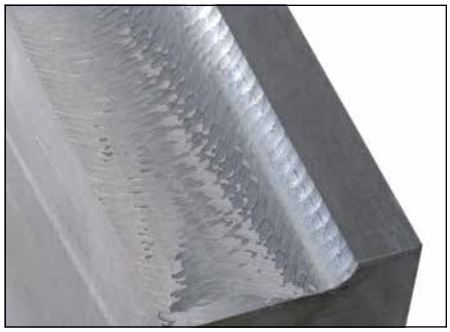
Each solution in itself offers decisive advantages in the machining process. But the combination of these TNC features, in particular, exploits the potential of the machine and tool and at the same time reduces the mechanical load. Changing machining conditions, such as interrupted cuts, various material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

– Active Chatter Control option (ACC)

dynamic + efficiency

Strong milling 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 antidote with the Active Chatter Control (ACC) control function. 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 (figure above) and with ACC (figure 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.

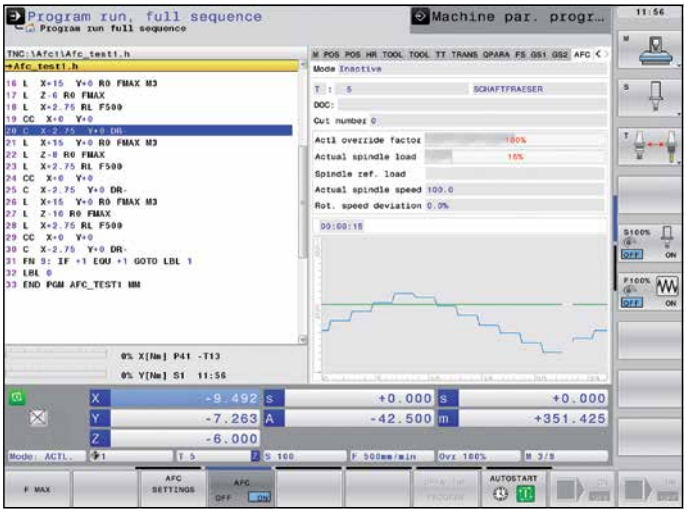
Adaptive feed rate control (AFC) automatically regulates the feed rate of the TNC, taking into consideration the respective spindle power and other process data. In a teach-in cut, the TNC records the maximum spindle power. Then, before actual machining, you define in a table the respective limit values between which the TNC can influence the feed rate in the "control" mode. Of course, various overload reactions can be provided for, which can also be defined by your machine tool builder.

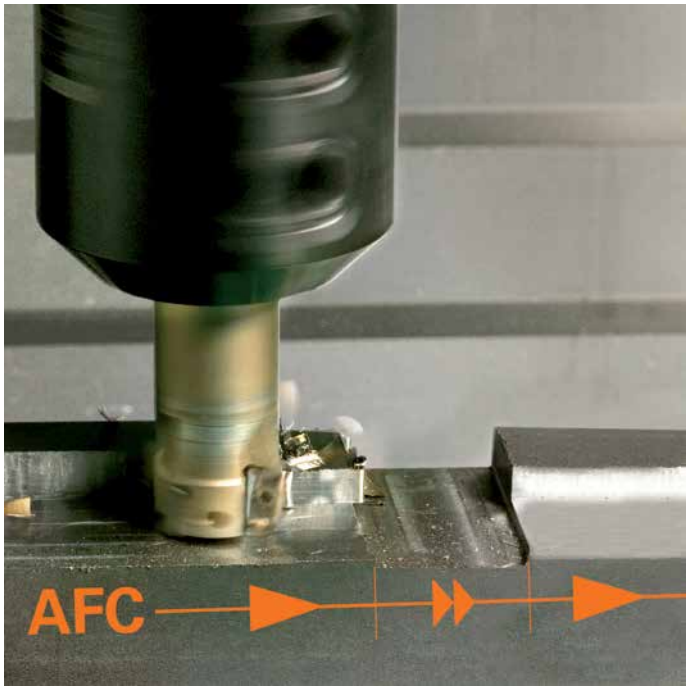
Adaptive feed rate control offers various advantages:

Optimizing the machining time
Fluctuations in dimensions or material (blowholes) can occur, particularly in 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 TNC'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 TNC reduces the feed rate. As soon as the feed rate falls below a defined minimum, the TNC 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.





Intelligent machining
– Machining any contour slots with trochoidal milling

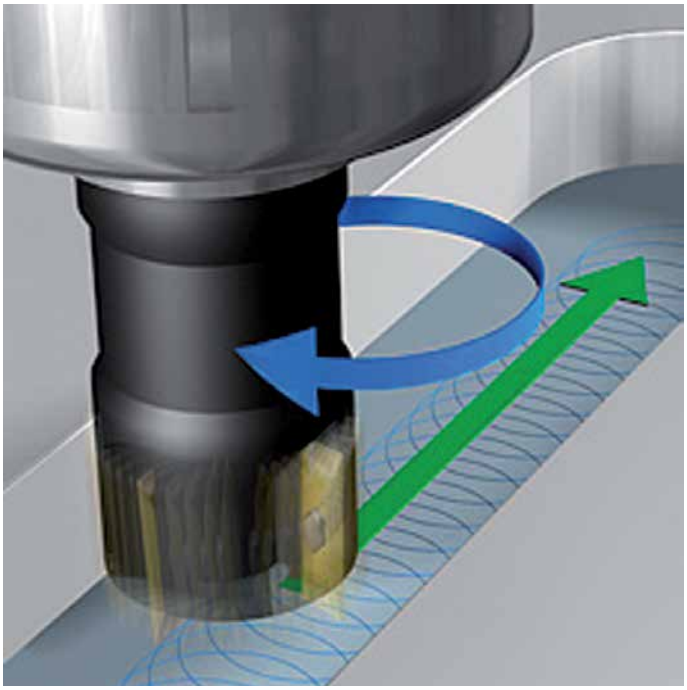
dynamic + efficiency

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
- Relieves mechanical load on the machine
- Less vibration
- Integrated finishing of the side wall



Minimize setup times
– The TNC 640 makes setup easy

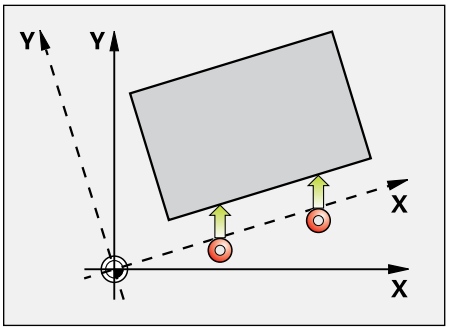
Before you can begin machining, you must first clamp the tool and set up the machine, find the position and orient the workpiece on the machine, and set the workpiece reference point. This is a time-consuming but indispensable procedure. After all, any error directly reduces the machining accuracy. Particularly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

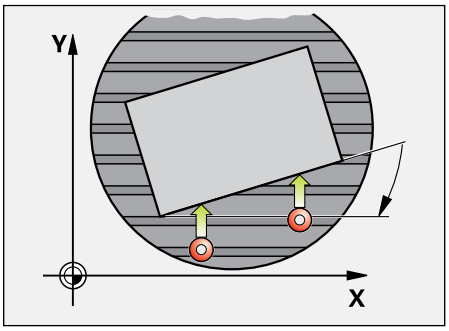
The TNC 640 features application-oriented, real-world setup functions. They support the user, help to reduce non-productive time, and make overnight, unattended production possible. Together with the **touch probes,** the TNC 640 offers numerous probing cycles for automatic alignment of the workpieces, 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 48). 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 45) 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 or by moving the rotary table.

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 TNC display to any workpiece position. Finding this point quickly and reliably reduces nonproductive time and increases machining accuracy.

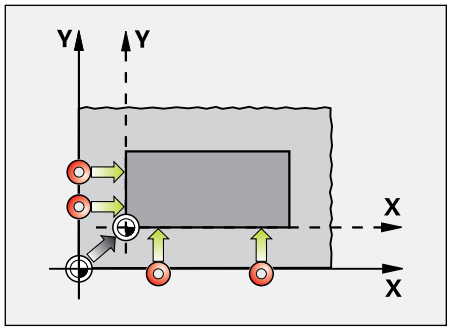
The TNC 640 features probing cycles for automatic presetting. Once found, you can save these datums
- in the datum management
- in a datum table, or
- by directly setting the displayed value.

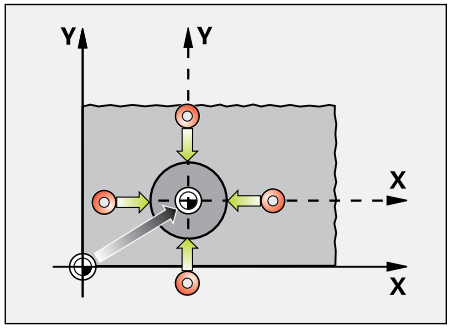
Datum management
The datum management makes flexible machining, shorter setup times and increased productivity possible. In other words, it makes it much easier to set up the machine.

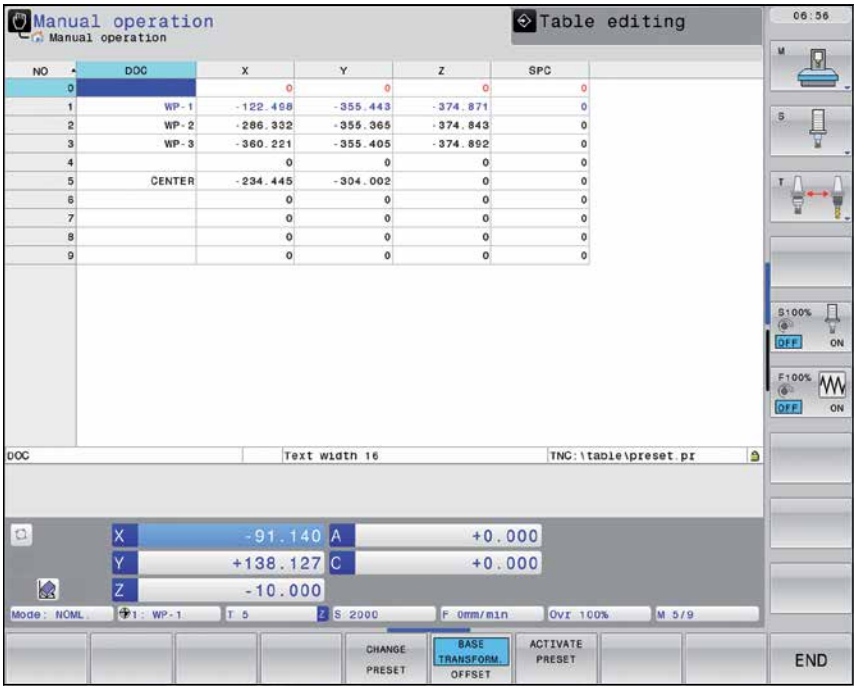
In the datum management you can save **any number of datums** and assign an individual basic rotation to each one.

There are three possibilities for rapid saving of the datums:
- In the Manual mode by soft key
- By using the probing functions
- With the automatic probing cycles

Set datum
At a corner, for example, or in the center of a circular stud







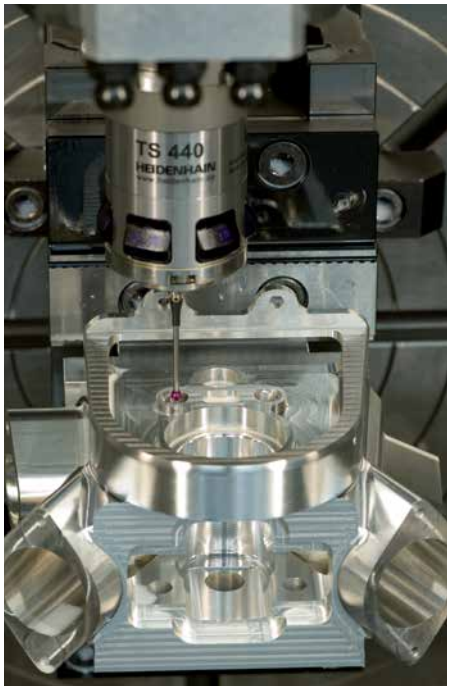
Automated machining
– The TNC 640 measures, manages and communicates

The difference in requirements placed on the classical machine for tool and mold-making and machining centers is 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 45) 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,
- ascertain the machining error trend.

Milling cutter measurement and automatic compensation of tool data
Together with the TT and TL touch probes for tool measurement (see page 46) 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.

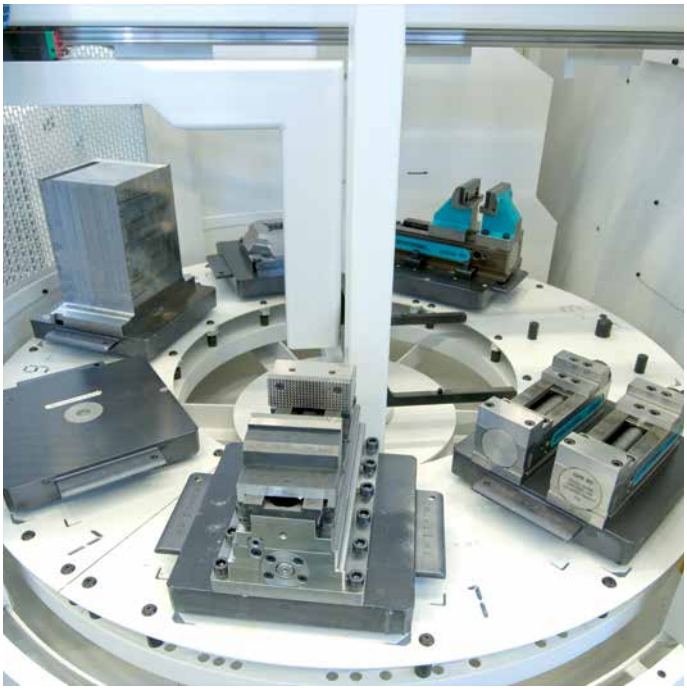


Tool management
For machining centers with automatic tool changers, the TNC 640 offers a central tool memory for any number of milling and turning tools. The tool memory is a freely configurable file and can therefore be optimally fitted to your needs. You can even have the TNC 640 manage your tool names. 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.

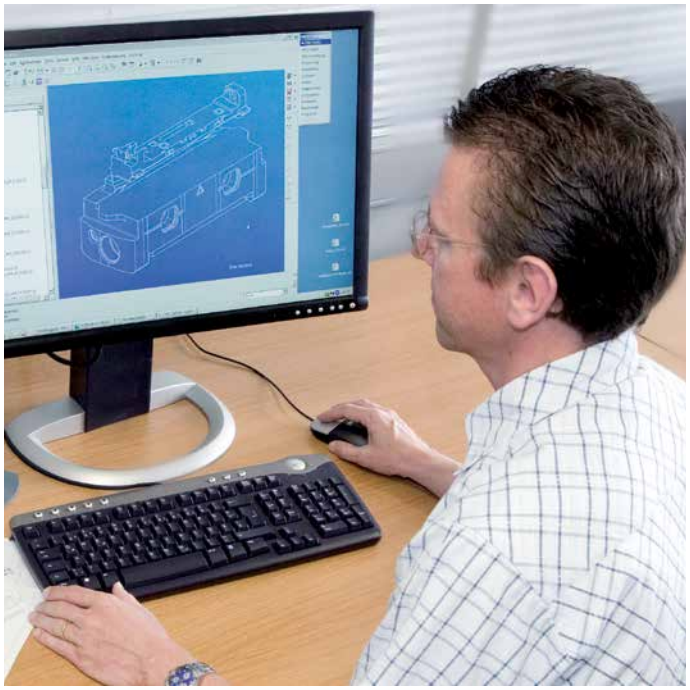
Positioning with Manual Data Input
You can start working with the TNC 640 even before writing a complete part program. Simply machine a part step by step—switching as you want between manual operation and automatic positioning.

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.

And if you are used to **G-code programming**, then the TNC is still the right control for you: you can use the G-code keyboard to enter the calculations, or enter them directly with the QWERTY keyboard.

Creating programs offline
The TNC 640 is also well equipped for offline programming. Through its interfaces it can be integrated into networks and connected with programming stations or other data storage devices.



– Graphic support in any situation

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.

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.

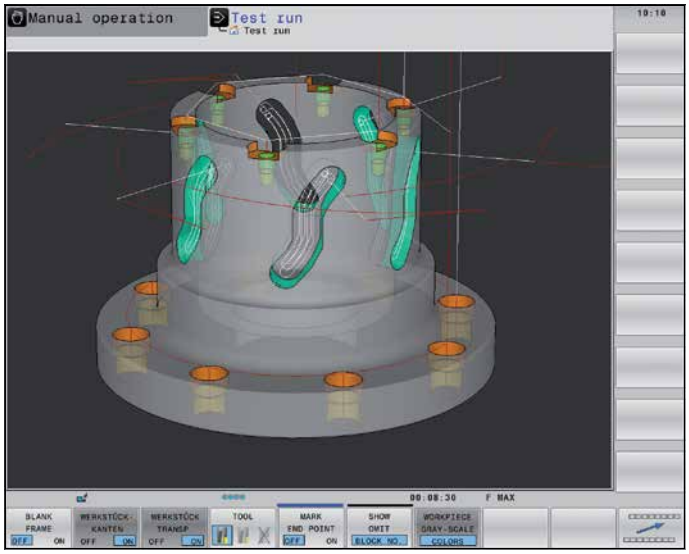
Program verification graphics
To be on the safe side before running a program, the TNC can graphically simulate the machining progress. The new finely detailed 3-D test graphics enables you to evaluate exactly the manufactured result even before the actual machining. You define the workpiece blank as a cuboid, cylinder or rotationally symmetric part with any contour. The TNC offers various ways to depict machining:

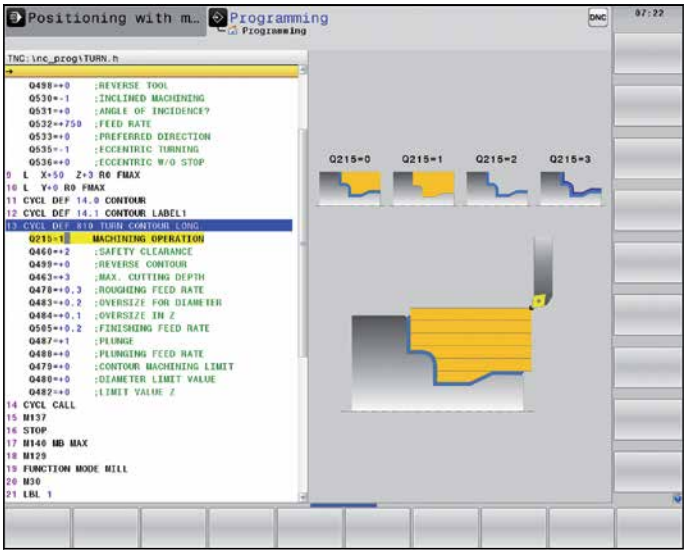
- Plan view with different shades of depth
- Three planes (as in the workpiece drawing)
- High-resolution 3-D view
- 3-D line graphics of the tool paths

You can adjust the type and quality of the image. Details can be displayed in magnification. In addition, the TNC 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.

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.





Programming in the workshop
– Straightforward function keys for complex contours

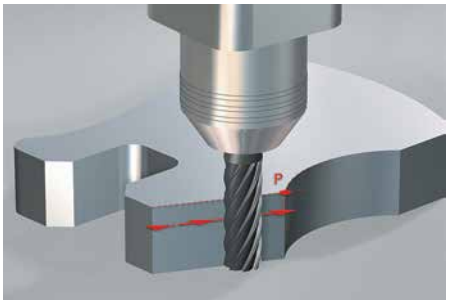
Programming 2-D contours

Two-dimensional contours are the bread and butter of the modern machine shop. 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.

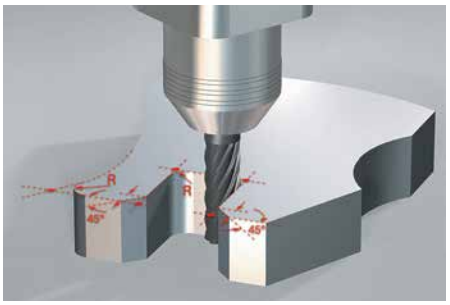
Programming with path function keys
If contours are dimensioned for NC, which means that the end points are specified in Cartesian or polar coordinates, then you can program them directly with the path function keys.

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, movement has to be smooth—that is, tangential.
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.

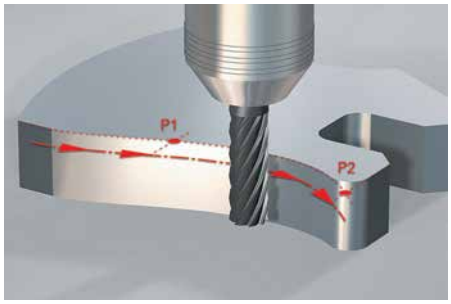


L
Straight line: Straight line defined by its end point

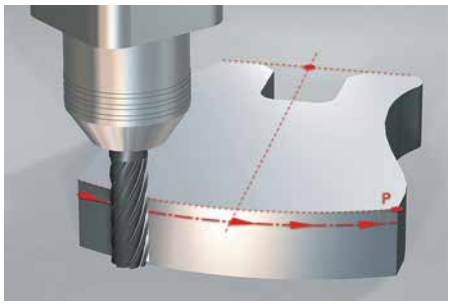


RND Corner rounding: Circular path defined by radius and corner point, with a smooth (tangen8tial) transition to its adjoining contour elements

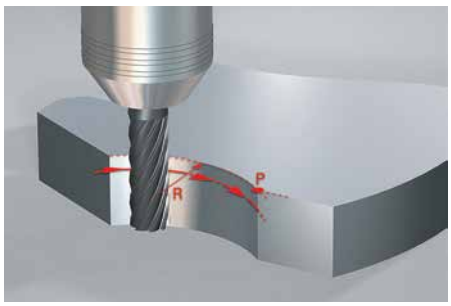
CHF Chamfer: Defined by the corner point and chamfer length



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



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



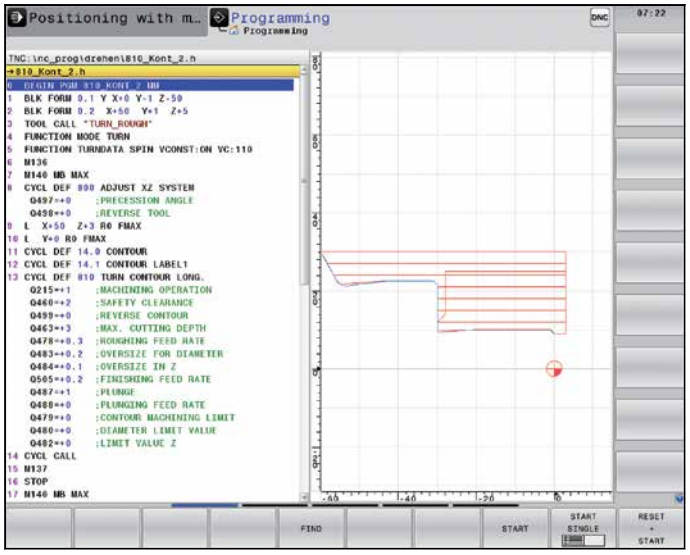
CR
Circular path defined by its radius, end point and rotational direction

– 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 memory as standard 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.

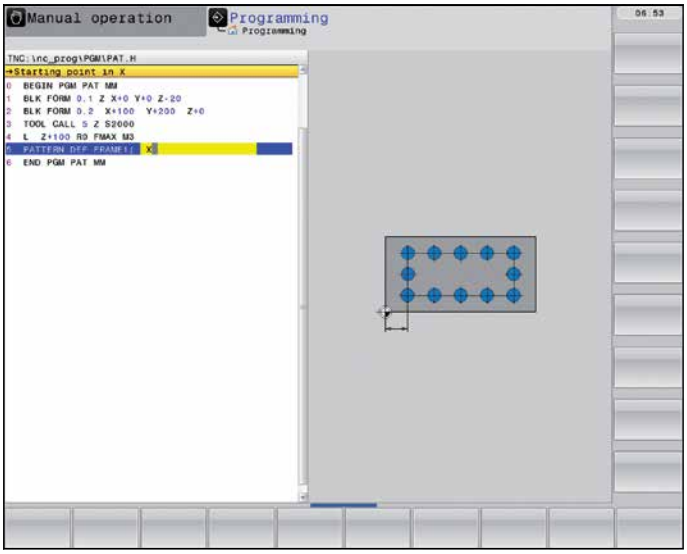
Up to twelve **subcontours** can be superimposed for machining. The control automatically calculates the resulting contour and the tool paths for roughing or clearing the surfaces. Subcontours can be pockets or islands. Different components are combined to form a single pocket in which the tool avoids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

The TNC 640 maintains a **finishing allowance** on the wall and floor surfaces during roughing. When **roughing** with different tools, the control recognizes the material remaining in inside corners so that it can clear it with smaller tools. A separate cycle is used for milling to the finished dimension.

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.

Simple and flexible programming of 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. 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.



Programming in the workshop
– Field-proven turning cycles (option)

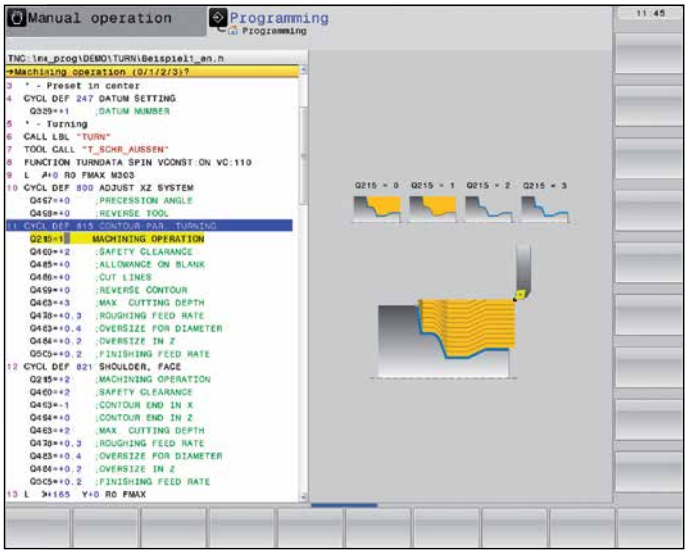
In the area of milling cycles, too, the TNC 640 offers a comprehensive and technologically ambitious package. They are equivalent to the proven and fully developed kernel functions of the HEIDENHAIN lathe controls. The user interface, however, is inspired in its look and functionality by the familiar 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 machined 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.



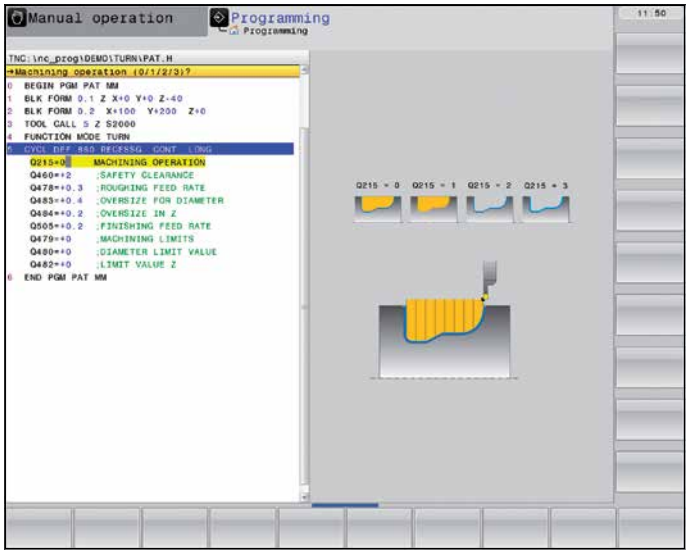
Recessing
In this area, as well, the TNC 640 distinguishes itself with ample flexibility and functionality. Simple recessing operations in longitudinal and transverse direction are just as possible as contour recessing, in which the cycle is machined along any desired contour. You can work particularly effectively during recess turning. Because infeed and cutting alternate directly, air cuts are hardly necessary. Here, too, the TNC considers the technological constraints (width of recessing tool from the tool table) and executes the operations quickly and reliably.

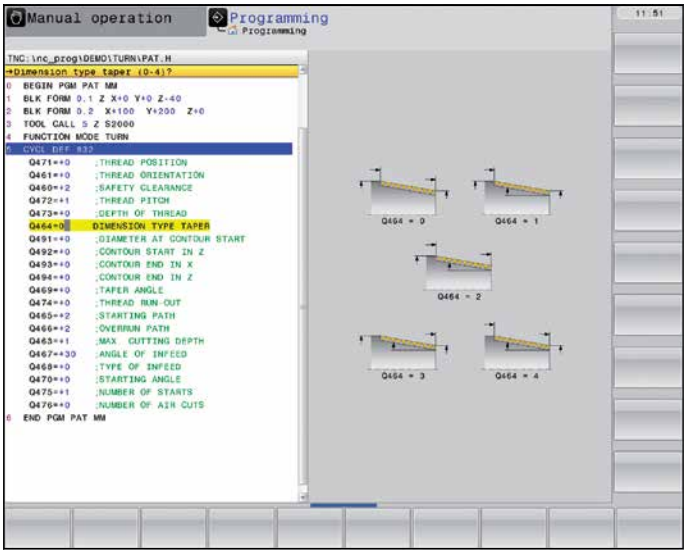
Thread machining
Simple and expanded cycles are available for longitudinal and transverse machining of cylindrical or tapered threads. You can use cycle parameters to define the manner in which the thread is produced. This enables you to machine a wide variety of materials.

Blank form update
Another highlight of the TNC 640 is the blank form update feature. If you define the workpiece blank at the beginning of your program, the control then computes the new blank for each following step. The machining cycles always adapt to the current workpiece blank. The blank for update feature helps you to avoid air cuts and optimize approach paths.

Orientation of the turning tool
On milling/turning machines it can be necessary to incline the tool during turning or change the side from which the part is to be machined. With the aid of a cycle, the TNC can change the tool's angle of incidence or 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.

Eccentric turning (option)
With the eccentric turning function you can perform turning operations even when the tool axis, due to the setup situation, is not aligned with the axis of rotation. During machining, the TNC 640 compensates any eccentricity with opposing movements of the linear axis coupled with the rotating spindle.





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.

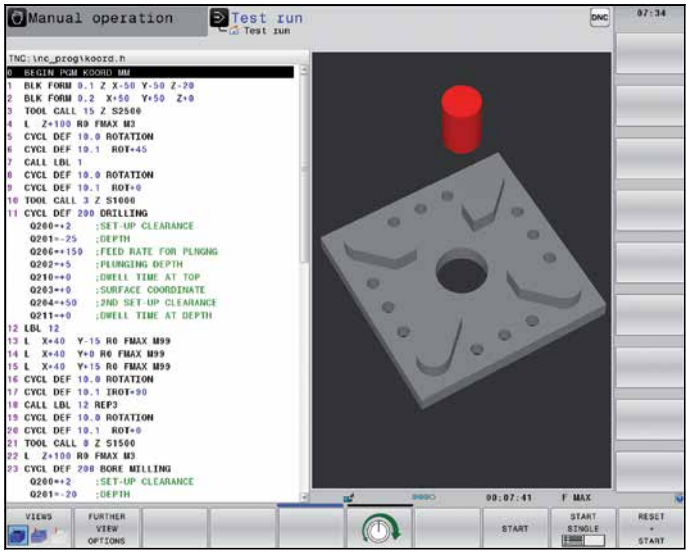
Program section repeats, subprograms, program calls
Many machining operations repeat themselves either on the same workpiece or on different workpieces. Once you have programmed a detail there is no reason to have to program it again. With its subprogramming feature, the TNC can save you a great deal 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 the **help system TNCguide,** a convenient help system that can display the user documentation in a separate window.

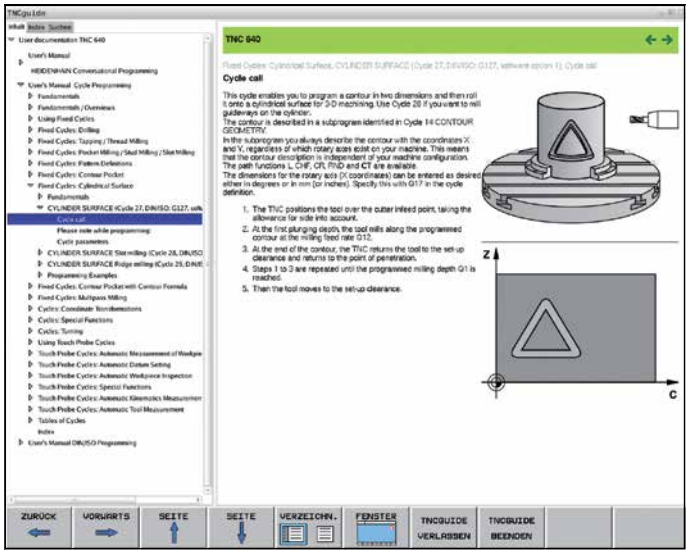
You can activate TNCguide by simply pressing the help key on the TNC keyboard or by clicking any soft key with a pointer in the shape of a question mark. You switch the cursor by simply clicking the help symbol that is always visible on the TNC screen.

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. This function is particularly helpful with the soft keys. The method and effect of operation are explained in detail.

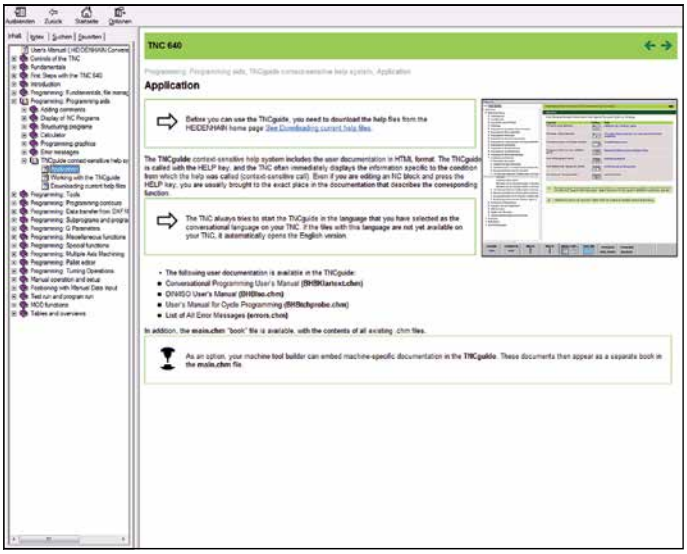
You can download the documentation in the desired language free of charge from the HEIDENHAIN homepage into the corresponding language directory on the TNC hard disk.

The following manuals are available in the help system:

- User's Manual for Conversational Programming
- User's Manual for Cycle Programming
- User's Manual for DIN/ISO Programming
- 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
– The TNC 640 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 TNC 640 in order to extract contours or machining positions from it. Not only does this save time otherwise spent on programming and testing, but you can also be sure that the finished contour is exactly according to the designer's specifications.

The DXF format—particularly the DXF format supported by the TNC 640—is very widespread, and is supported by all common CAD and graphics programs.

After the DXF file has been loaded onto the TNC from the network or a USB stick, you can open the file in the file manager of the TNC, just like an NC program.

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

The TNC also supports you when **defining the workpiece preset.** The datum of the drawing for a DXF file is not always located where you can use it directly as the workpiece preset, especially when the drawing contains multiple views. For this reason, the TNC 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 mid-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

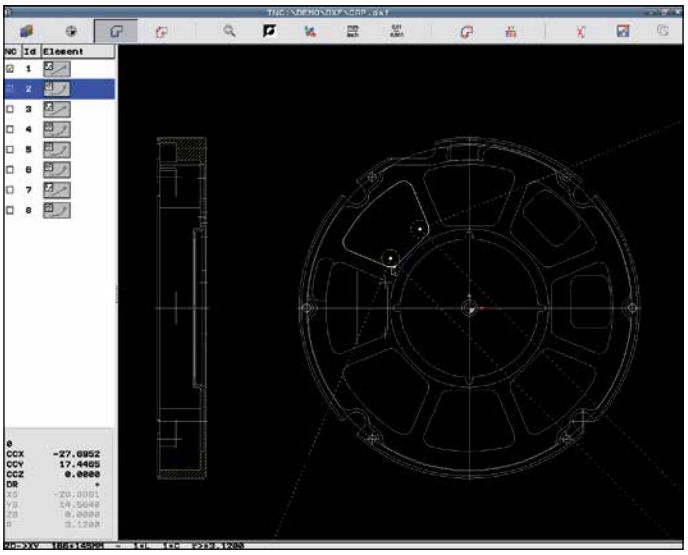
If multiple intersections result between two elements (e.g., between a straight line and a circle), you can select the correct intersection with a mouse click.

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 TNC detects your desired direction of machining, and starts the **automatic contour detection.** The TNC 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.

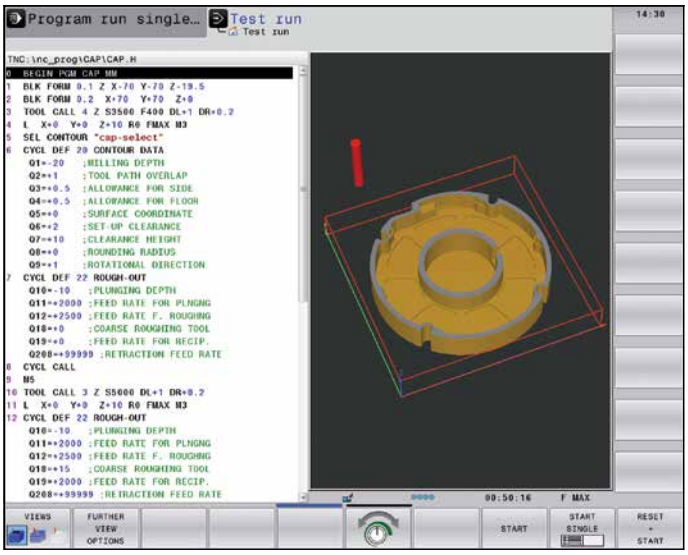
But you can also select **machining positions** and save them as point files, particularly in order to use drilling positions or starting points for pocket machining.

This can be done very easily: Using the mouse, 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. Moreover, you can define the resolution of the contour program to be uploaded in case you want to use it on older TNC controls, or a transition tolerance if occasionally the elements do not quite adjoin.



Contour selection from an imported DXF file



Part program on the basis of the imported DXF file

Open for communication
– Fast data transfer with the TNC

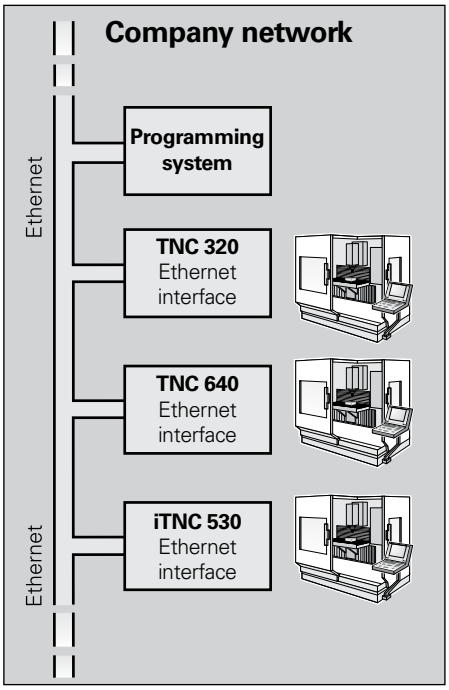
The networked TNC 640
The TNC 640 can be integrated into networks and connected with PCs, programming stations and other data storage devices. Even in its standard version, the TNC 640 features a latest-generation Gigabit Ethernet interface in addition to its RS-232-C/V.24 data interface. The TNC 640 communicates with NFS servers and Windows networks in TCP/IP protocol without needing additional software. The fast data transfer at rates of up to 1000 Mbit/s guarantees very short transfer times.

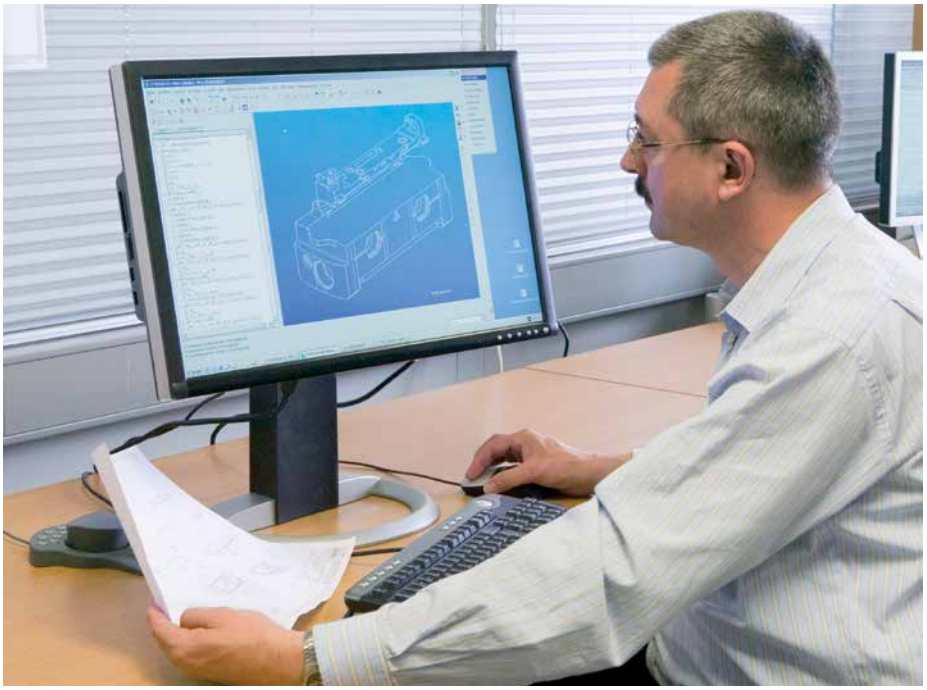
The transmitted programs are saved in the internal memory of the TNC 640 and are run from it at high speed.

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 software **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,
- start 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.





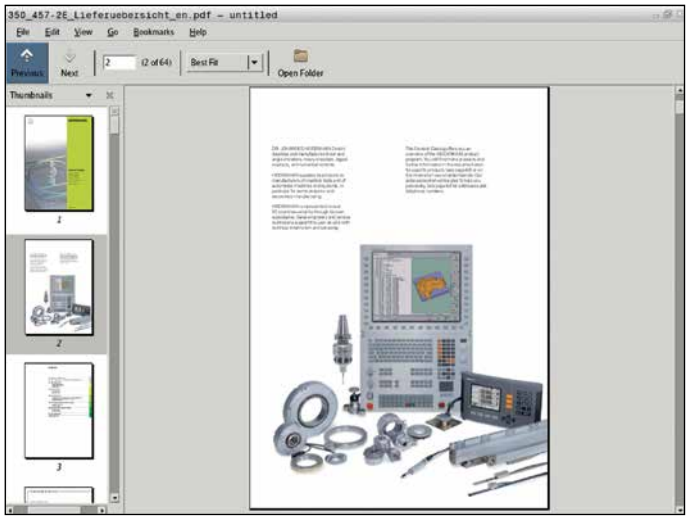
– Display any file formats on the TNC screen

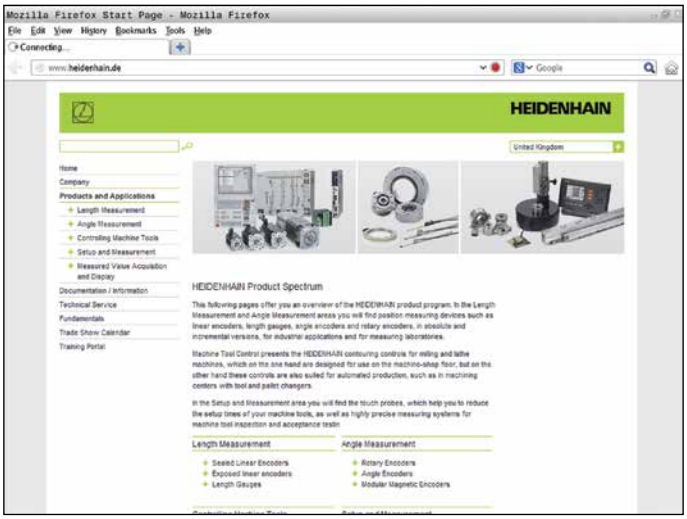
The new integrated PDF viewer enables the user to open PDF files directly on the control. The PDF format is a widely used data format that can be generated out of a great variety of applications. This enables you to easily view work instructions, drawings or other information in the TNC.

The integrated browser now lets you connect the TNC to the Internet and access it directly from the control.

The following further file formats can also be opened directly on the TNC with corresponding editors, and sometimes edited.

- Text files ending with .txt or .ini
- Graphic files ending with .gif, .bmp, .jpg, .png
- Table files ending with .xls or .csv
- HTML files





Open for communication
– 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: At 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.

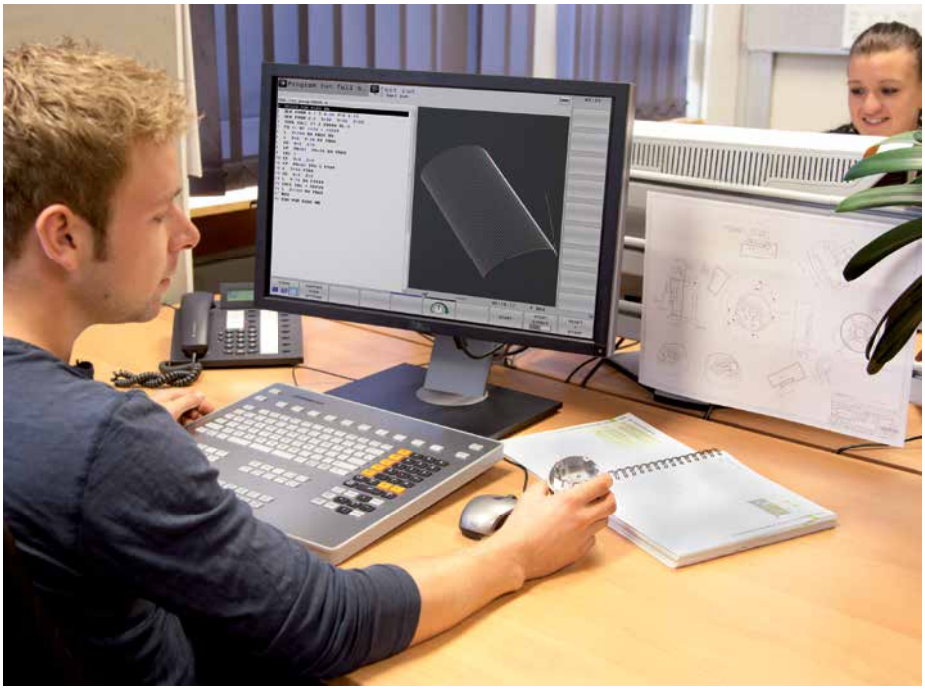
Training with the programming station
Because the TNC 640 programming station is based on the same software as the TNC 640, it is ideally suited for apprentice and advanced training. The program is entered on the original keyboard unit. Even the test run functions exactly as it does on the machine. This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

Because it can be programmed in HEIDENHAIN conversational language and in DIN/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

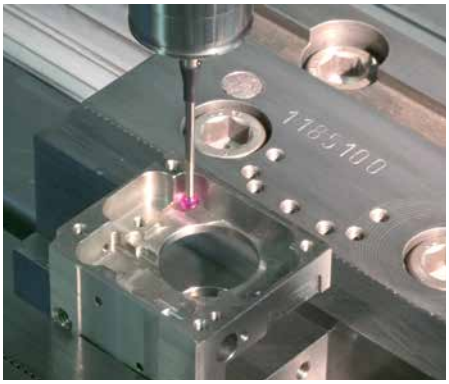
Workpiece touch probes from HEIDENHAIN help you to reduce costs in the workshop and in series production: Together with the TNC 640, touch probes can automatically perform setup, measuring and inspection functions.

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 probes with **cable connection for signal transmission** for machines with manual tool change:
TS 230 – HTL version
Touch probes with **infrared signal transmission** for machines with automatic tool change:
TS 440 – Compact dimensions
TS 444 – Compact dimensions, battery-free power supply through integrated air turbine generator over central compressed air supply
TS 640 – Standard touch probe with wide-range infrared transmission
TS 740 – High probing accuracy and repeatability, low probing force



TS 640

TS 440

SE 640

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

Measurement of milling cutters
– 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 either before machining or during interruptions.

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

The **TT 140** uses signal transmission by cable, whereas the **TT 449** operates with signal transmission over infrared beam and does not require a cable. 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 deviations of individual teeth along with the tool length and radius.

TT 449

TL Micro

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

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.

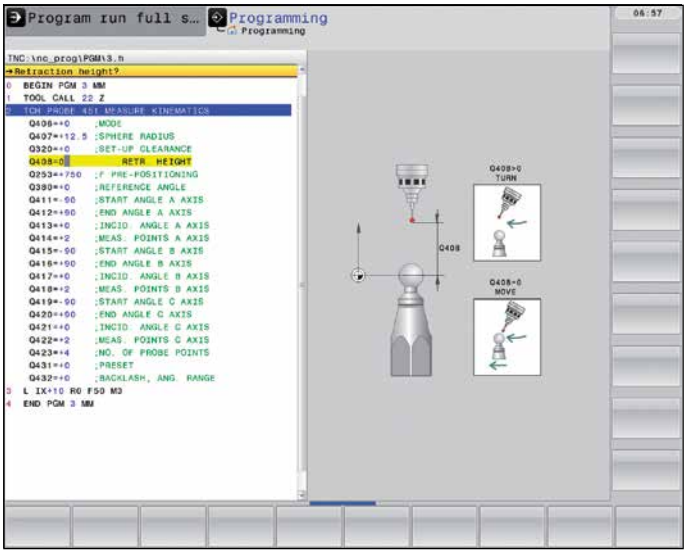
The new TNC function **KinematicsOpt** is an important component to help you meet these high requirements: With a HEIDENHAIN touch probe inserted, a 3-D touch probe cycle measures your machine's rotary axes fully automatically. The results of measurement are the same regardless of whether the axis is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

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. 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.

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

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	•		HEIDENHAIN conversational
	•		DIN/ISO programming
		42	Direct loading of contours or machining positions from DXF files and saving as conversational contouring program or as point table
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
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	•		Automatic calculation of spindle speed, cutting speed, feed per tooth and 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 highly smoothed jerk
		9	3-D tool compensation through surface normal vectors
		9	Tool Center Point Management (TCPM): Using the electronic handwheel to change the angle of the swivel head during program run without affecting the position of the tool point
		9	Keeping the tool normal to the contour
		9	Tool radius compensation normal to the tool direction
		9	Manual traverse in the active tool-axis system
Rotary table machining		8	Programming of cylindrical contours as if in two axes
		8	Feed rate in distance per minute
Turning		50	Program-controlled switchover between milling and turning
		50	Constant surface speed
		50	Tool-tip radius compensation
		50	Cycles for roughing, finishing, recessing, thread turning and recess turning
		50	Updating of the workpiece blank during contour cycles
		50	Turning-specific contour elements for recesses and undercuts
		50	Orientation of the turning tool for outside or inside machining
		50	Inclined turning
		50	Speed limiting
		50	Eccentric turning (additionally required: option 135)

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
		50	Recess
		50	Undercut
Approaching and departing the contour	•		Via straight line: tangential or perpendicular
	•		Via circular arc
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
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
		50	Area clearance cycles, longitudinal and transverse, paraxial and contour parallel
		50	Recessing cycles, radial/axial
		50	Radial/axial recess turning cycles (combined recessing and roughing motion)
	•		Milling internal and external threads
		50	Turning inside and outside 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, contour pocket
	•		Contour slot with trochoidal milling
	•		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	•		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, arcus sin, arcus cos, arcus 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

User functions	Default	Option	
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 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
Program verification 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
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
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 Program interruption, contour departure and return
Datum management	•		One table for storing any datums
Datum tables	•		Several datum tables for storing workpiece-related datums
Pallet tables	•		Pallet tables (with as many entries as desired for the selection of pallets, NC programs and datums) can be machined workpiece by workpiece
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 Automatic measurement and optimization of machine kinematics
Parallel secondary axes	• • •		Compensating movement in the secondary axes U, V, W through the principal axes X, Y, Z Including movements of parallel axes in the position display of the associated principal axis (sum display) Defining the principal and secondary axes in the NC program makes it possible to run programs on different machine configurations
Conversational languages	•		English, German, Chinese (traditional, simplified), Czech, Danish, Dutch, Finnish, French, Hungarian, Italian, Korean, Norwegian, Polish, Portuguese, Romanian, Russian (Cyrillic), Spanish, Swedish, Slovenian, Slovak, Turkish

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 230:** 3-D touch trigger probe with cable connection or • **TS 440:** 3-D touch trigger probe with infrared transmission or • **TS 444:** 3-D touch trigger probe with infrared transmission or • **TS 640:** 3-D touch trigger probe with infrared transmission or • **TS 740:** 3-D touch trigger probe with infrared transmission
Tool measurement	• **TT 140:** 3-D touch trigger probe or • **TS 449:** 3-D touch trigger probe with infrared transmission • **TL Nano:** Laser system for contact-free measurement of tools or • **TL Micro:** Laser system for contact-free measurement of tools
Programming station	Control software for PCs for programming, archiving, and training • Single-station license with original control keyboard • Single-station license with virtual keyboard • Network license with virtual keyboard • Demo version (operated via PC keyboard—free of charge)
Software for PCs	• **TeleService:** Software for remote diagnostics, 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 34059x-	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	617920-01	**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 transformation:** Tilting the working plane, PLANE function
9	Software option 2	01	617921-01	**Interpolation:** Linear in 5 axes **3-D machining** • 3-D tool compensation through surface normal vectors • Tool center point management (TCPM): Using the electronic handwheel to change the angle of the swivel head during program run without affecting the position of the tool point • Keeping the tool normal to the contour • Tool radius compensation normal to the tool direction • Manual traverse in the active tool-axis system
18	HEIDENHAIN DNC	01	526451-01	Communication with external PC applications over COM component

Option number	Option	As of NC software 34059x-	ID	Remark
23	Display step	01	632986-01	**Display step** to 0.01 µm or 0.00001°
40	DCM collision	02	526452-01	Dynamic Collision Monitoring (DCM)
42	DXF converter	02	526450-01	Load and convert DXF contours
45	Adaptive Feed Control (AFC)	02	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
50	Turning	01	634608-01	Turning functions: • Tool management for turning • Tool-tip radius compensation • Switching between milling and turning modes of operation • Lathe-specific contour elements • Package of turning cycles
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	676938-01	Extended tool management
133	Remote desk. manager	01	894423-01	Display and remote operation of external computer units (e.g. a Windows PC)
135	Synchronizing functions	04	1085731-01	RTC: Real-Time Coupling function for synchronizing axes and spindles
141	Cross talk comp.	02	800542-01	CTC: Compensation of axis coupling
142	Position adapt. control	02	800544-01	PAC: Position-dependent adaptation of the controller parameters
143	Load adapt. control	02	800545-01	LAC: Load-dependent adaptation of the controller parameters
144	Motion adapt. control	02	800546-01	MAC: Motion-dependent adaptation of control parameters
145	Active chatter control	02	800547-01	ACC: Active Chatter Control
146	Active vibration damping	04	800548-01	AVD: Active vibration damping

Overview
– Specifications

Specifications	Default	Option	
Components	•		MC main computer
	•		CC or UEC controller units
	•		BF color TFT flat-panel display with soft keys (15.1 or 19 inches)
	•		TE operating panel (suitable for screens with 15.1 or 19 inches)
Operating system	•		HEROS 5 real-time operating system for machine control
Memory	•		Random access memory (RAM): 4 GB
	•		Program memory: SSDR approx. 21 GByte HDR approx. 144 GByte
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 (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
	•		Helix: Superimpositioning 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/1024
	•		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: minimum 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 Gigabit Ethernet interface 1000BASE-T
	•		5 x USB (1 x front USB 2.0, 4 x 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: 5 °C to 40 °C
	•		Storage: -20 °C to +60 °C

– Comparison of controls

Comparison of controls	**TNC 620** NC SW 81760x-01	**TNC 640** NC SW 34059x-04	**iTNC 530** NC software 60642x-03
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)	–	●	●
• Milling/turning operation (up to 18 axes + 2 spindles)	–	**Option**	–
Program entry			
• HEIDENHAIN conversational format	●	●	●
• According to ISO	●	●	●
• DXF converter	Option	Option	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
Global program settings	–	★	Option
KinematicsOpt	Option	Option	Option
Touch probe cycles	Option	●	●
Pallet editor	Option	●	●
Parallel-axis function	●	●	–

TNC 620

TNC 640

iTNC 530

● Function available
★ Function planned

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
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**
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

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

MY **ISOSERVE SDN. BHD.**
43200 Balakong, Selangor
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**
02-384 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.**
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.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!*

892916-24 · 7 · 3/2014 · H · Printed in Germany