HEIDENHAIN

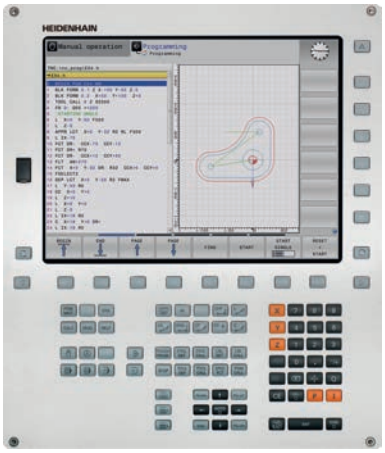
TNC 620

The Compact Contouring Control for Milling, Drilling, and Boring Machines

HSCI

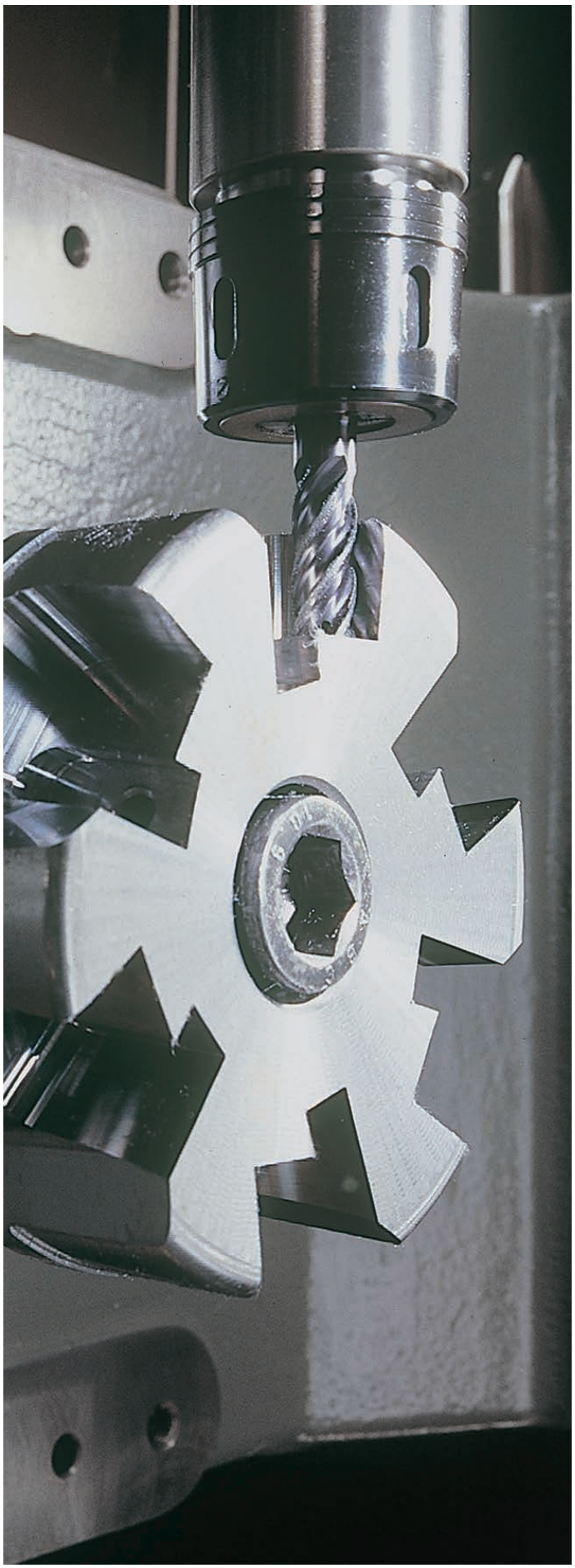
04/2019

TNC 620
with touchscreen



TNC 620
with operating keys

TNC 620
with operating keys and alphanumeric keyboard



The functions and specifications described in this brochure apply to the TNC 620 with NC SW 81760x-06.

Contents

The TNC 620...

Compact and versatile
– The right control for milling, drilling, and boring 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 on machining centers. During this period the controls have been continuously developed with the needs of the machine operator always placed foremost.

These principles can also be found in the TNC 620: workshop-oriented programmability with graphical support, many practical cycles, and an operational design that you are familiar with from other HEIDENHAIN controls.

The TNC 620 is compact and easy to read. The TNC 620 is a compact, versatile contouring control with up to five controlled axes and controlled spindle. Thanks to its user-friendly operation and scope of features, it is especially well suited for use on universal milling, drilling, and boring machines for:

- Series and single-part production
- Tool making
- Machine building
- Research and development
- Prototypes and pilot plants
- Repair departments
- Training and education facilities

Multitouch operation
The TNC 620 is available not only in the conventional version with a standard screen and keyboard, but also with a touchscreen and keyboard.

Whether zooming with two fingers, rotating, or moving, you operate the TNC 620 quickly and easily with your fingertips.

Shop-oriented programming
You program conventional milling and drilling operations yourself at the machine, in Klartext—the dialog-guided, workshop-oriented programming language from HEIDENHAIN. The TNC 620 provides you with optimum support with practical prompts, questions, and graphical aids.

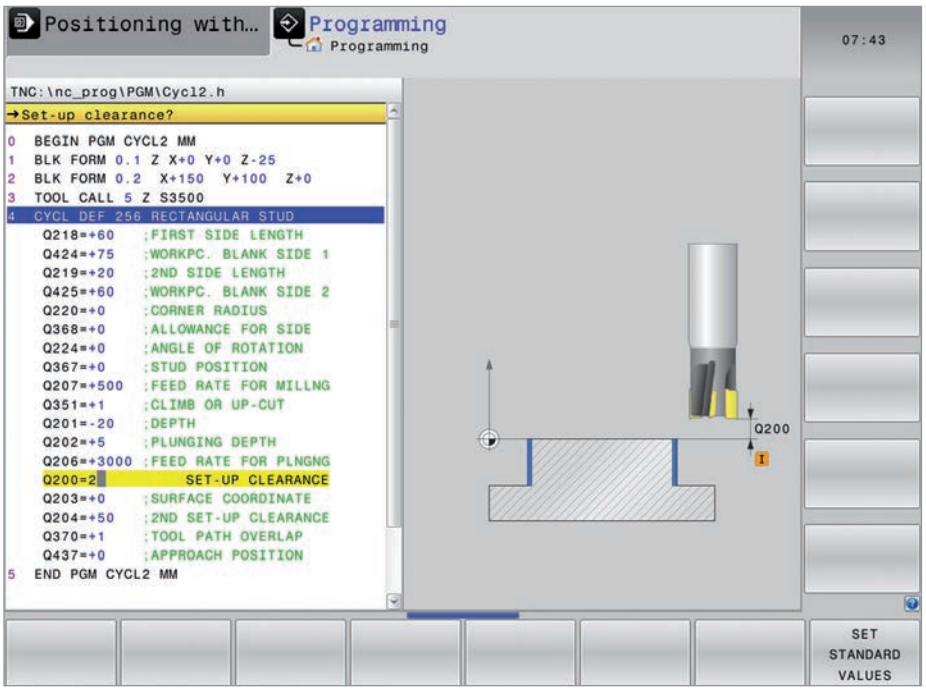
Standard operations and even complex applications are on call as a large variety of real-world machining cycles or coordinate transformations.

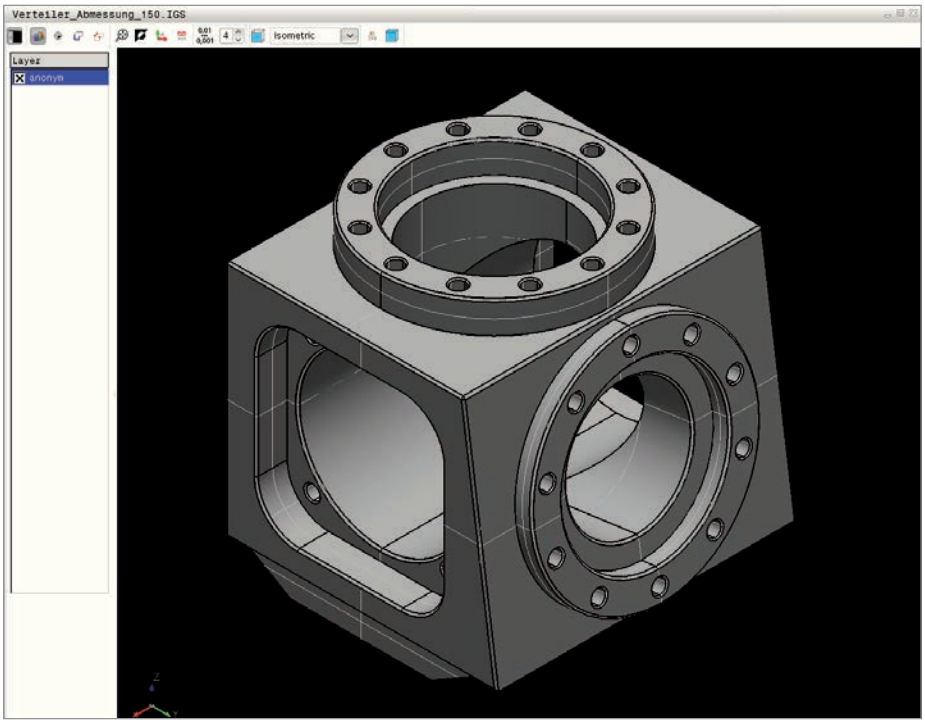
Offline program creation
The TNC 620 can be programmed remotely just as well. Its Fast Ethernet interface guarantees very short transfer times, even of long programs. In addition, data can be transferred quickly and easily via the USB interface.

CAD viewer as standard
Display your CAD data in the .step or .iges format directly on the TNC 620. The standard integrated CAD viewer can open many common 3-D formats and display them on the TNC 620 screen.

Finely detailed graphics
Thanks to the high-resolution program verification graphics of the TNC 620, you can exactly evaluate the result of milling or drilling processes even before the actual machining operation.

Advanced Dynamic Prediction (ADP)
The TNC 620 offers its ADP function for demanding milling operations. This optimizes the motion control for all feed axes. With the TNC 620, you achieve even higher surface quality and contour fidelity.





Intuitive and user-friendly
– Modern multitouch operation

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

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

During the program run, status displays are always available on the screen, informing you about the tool position, the current program, the active cycles, and coordinate transformations, etc. The TNC 620 even shows the current machining time.

Well-thought-out touchscreen
The operation of the TNC with its easy-to-read control panel has proved itself over many years. Operators around the world use their TNC with dialog keys, navigation keys, and soft keys.

In a touchscreen version, the TNC 620 now supports you with a particularly innovative and user-friendly operating interface. It combines the proven advantages of the HEIDENHAIN controls with a new type of operation by tapping, swiping, and dragging.

Easy to operate
The TNC 620 can also be conveniently operated through a connected mouse. In the workshop, however, it's sometimes hard to find an adequate running surface for a mouse. With a touchscreen, you do not need additional workspace for input devices. In addition, operating the control is even easier: swiping, direct selection of controls, and navigation in menus make daily work on your TNC 620 easier. The CAD importing is especially convenient with the touchscreen. In drawings, you can zoom, move, or select quickly and easily by gesture.

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

PLC functions keys (soft keys) for machine functions

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

Axis keys, numeric keypad, and **navigation**

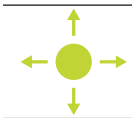
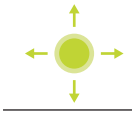
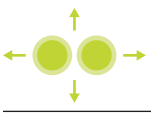
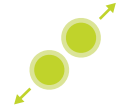
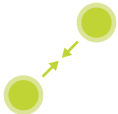
Shortcut menu

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

Machine operating panel with control keys and override potentiometers for feed rate and spindle speed

Practical touchscreen
The touchscreen was conceived for harsh workshop conditions. It is splash-proof, scratch-resistant, and certified for IP54 protection. If you want to clean your screen, you can simply select "Touchscreen Cleaning" mode. This locks the screen to prevent unintended operation.

Gestures for multitouch operation
The screen of the TNC 620 can be operated with gestures such as those commonly used on mobile devices. For example, you can zoom in or out with two fingers. By swiping, you can very quickly navigate in soft-key rows, programs, or menus.

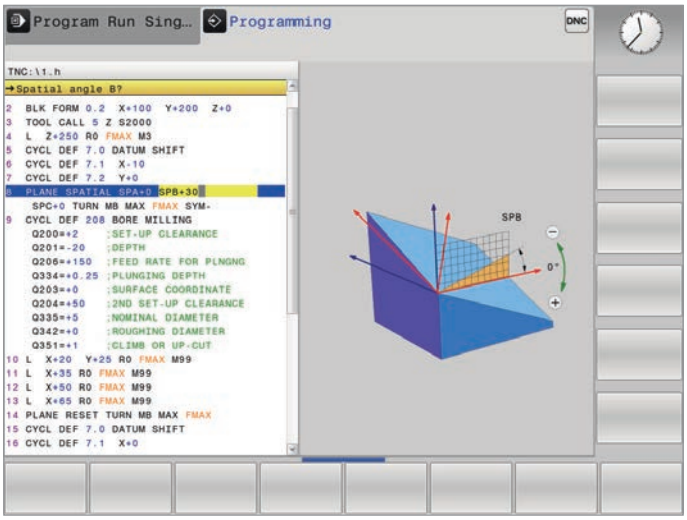
Symbol	Gesture
	Tap
	Double tap
	Long press
	Swipe
	Drag
	Two-finger drag
	Spread
	Pinch

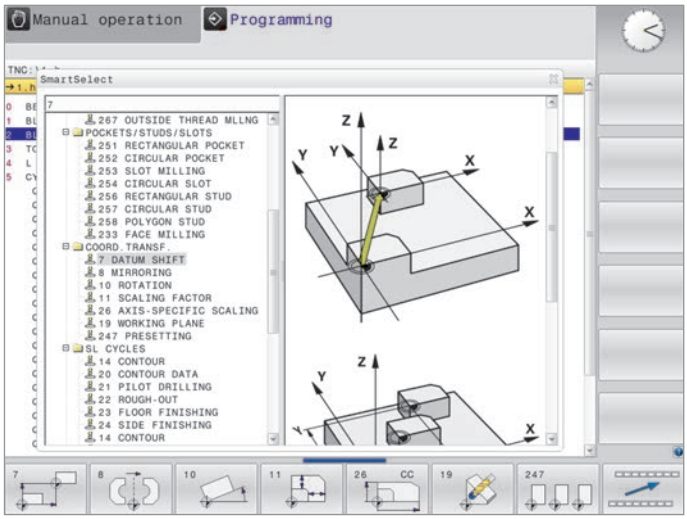
Optimal screen display
The user interface of the TNC 620 has a modern appearance, with lightly rounded forms, color gradients, and a homogeneously designed font. The individual screen areas are clearly distinguished and the operating modes are also indicated by their respective symbols. To better distinguish between the priority of error messages, the TNC 620 displays them in color-coded categories. A color-coded warning triangle is also displayed.

Intuitive 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—principles that HEIDENHAIN has always represented. However, the TNC 620 also offers a number of features that make working with the control even easier and user-friendlier than ever.

Fast function overview
With **smartSelect** you enjoy dialog guidance for quick and easy selection of functions 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-hand 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. Also, smartSelect enables you to define fixed cycles, touch probe cycles, and special functions (SPEC FCT), and quickly access the parameter programming.





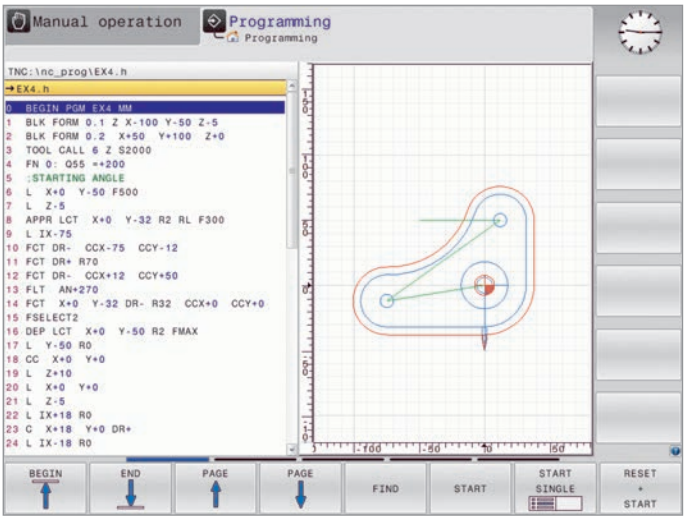
Color-structured programs
The content of a program line can be quite comprehensive: line number, program function, input values, comment. To help you always find your way even in complex programs, the individual program elements on the TNC 620 are shown in different colors. The color syntax highlighting improves your overview when editing NC programs. It enables you to see at a glance, for example, where the editable input values are.

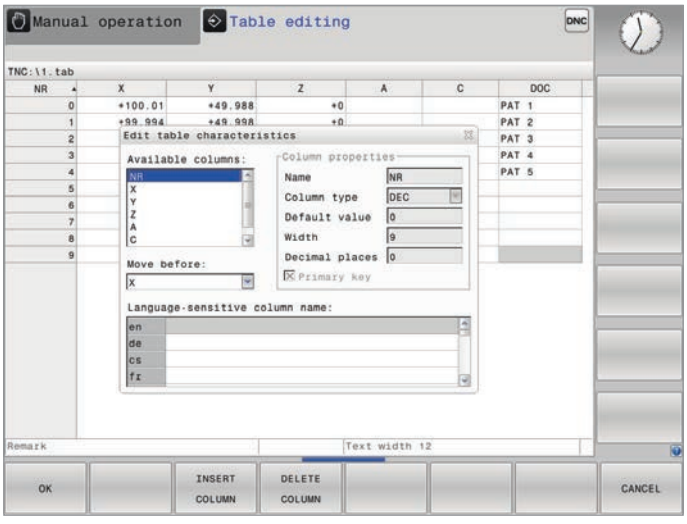
Uniform table editor
Regardless of which table you are editing—whether the tool table, preset table, or pallet table—the appearance, function, and operation of the table editor are always the same.

Info line
In the info line, the TNC 620 shows the respective submode condition and helps you to orient yourself. The function is comparable to 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.

User administration
The user administration of the TNC 620 lets you specify various roles and access rights for users. Each user can then operate only within his assigned rights. This can prevent an unintentional or unauthorized deleting of files or contents of system files. Additionally, many functions can be accessed only if the user has the corresponding rights. The user administration of the TNC 620 thus not only provides increased data protection, but also increases machine operating safety.



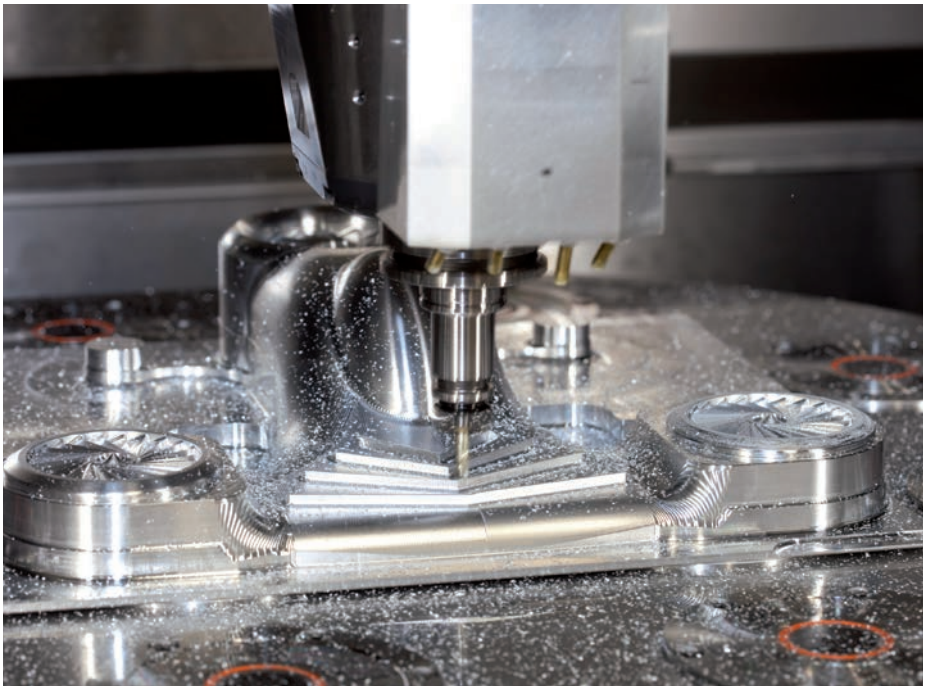


Quick and reliable machining with high contour fidelity
– The TNC 620 permits optimum tool movement

Very high contour fidelity and surface quality
TNC controls from HEIDENHAIN are known for their **jerk-smoothed as well as velocity- and acceleration-optimized motion control.** In this way, you can ensure optimized surface quality and workpiece accuracy. With the TNC 620 you can exploit state-of-the-art developments. The TNC 620 looks ahead, thinks along with you, and can calculate the contour dynamically before machining. Special filters specifically and additionally suppress machine-specific natural vibrations.

With **look-ahead**, the TNC 620 recognizes directional changes beforehand and adapts the traversing speed to the course of the contour and the surface to be machined. You simply program the maximum machining velocity as feed rate and, in **Cycle 32 TOLERANCE**, enter in the control the maximum permissible deviations from the ideal contour. The TNC 620 automatically adapts the machining to the tolerance that you define. No contour damage occurs with this method.

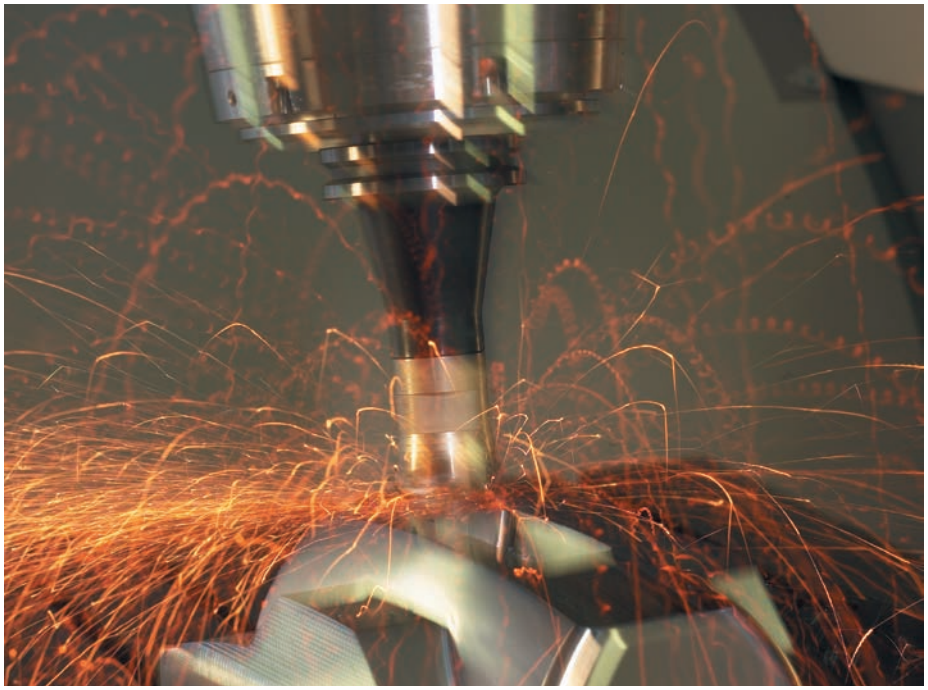
Advanced Dynamic Prediction (ADP) expands the previous advance calculation of the permissible maximum feed rate profile. ADP compensates differences in feed rate profiles resulting from point distribution on neighboring paths, especially in NC programs from CAM systems. This provides, among other things, a particularly symmetric feed rate behavior on the back-and-forth paths during bidirectional finish milling, as well as very smooth feed rate curves on parallel milling paths.



Fast machining and computing processes
The fast block-processing time of maximum 1.5 ms enables the TNC 620 to run fast advance calculations in order to optimally use the dynamic parameters of the machine. In this way, functions like ADP and look-ahead not only provide very high contour accuracy and surface definition—they also optimize the machining time.

One of the reasons for the TNC 620's high speed is its **uniformly digital control design.** It consists, on the one hand, of the integrated digital drive technology from HEIDENHAIN, and on the other hand all control components are interconnected with digital interfaces—the control components via HSCI (HEIDENHAIN Serial Controller Interface), and the encoders via EnDat 2.2. This makes it possible to realize very high feed rates. While interpolating simultaneously in up to five axes, the TNC 620 attains the required cutting speeds by digitally controlling spindle speeds up to **100 000 rpm**.

The TNC 620's powerful 5-axis machining enables you to manufacture even complex 3-D contours economically. The required programs are usually created on external CAM systems and comprise a large number of short line segments that are transferred to the control. With its short block-processing time, the TNC 620 quickly executes even complex NC programs. Thanks to its computing power, however, it can also transfer complex advance calculations to simpler NC programs. This makes it unimportant what data volume the NC programs from your CAD systems have: with the TNC 620, the finished workpiece will be a virtually perfect reflection of the generated program.

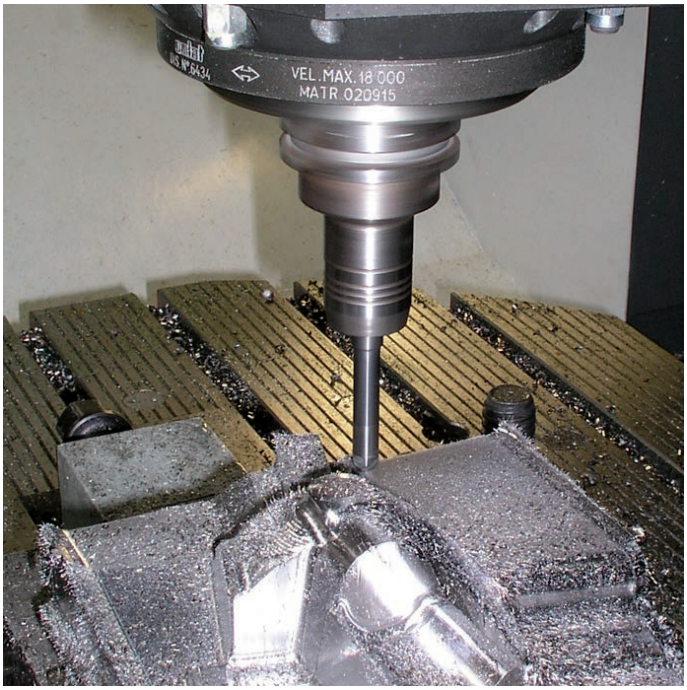


Quick and reliable machining with high contour fidelity
– Dynamic Precision

dynamic + precision

The control design of the TNC 620 guarantees high accuracy and surface definition at high machining speeds. These are made possible by various technologies, cycles, and functions. Individually or in combination, they ensure optimized motion control, effective jerk limiting, and dynamic contour look-ahead, and therefore perfect surfaces with very short machining times.

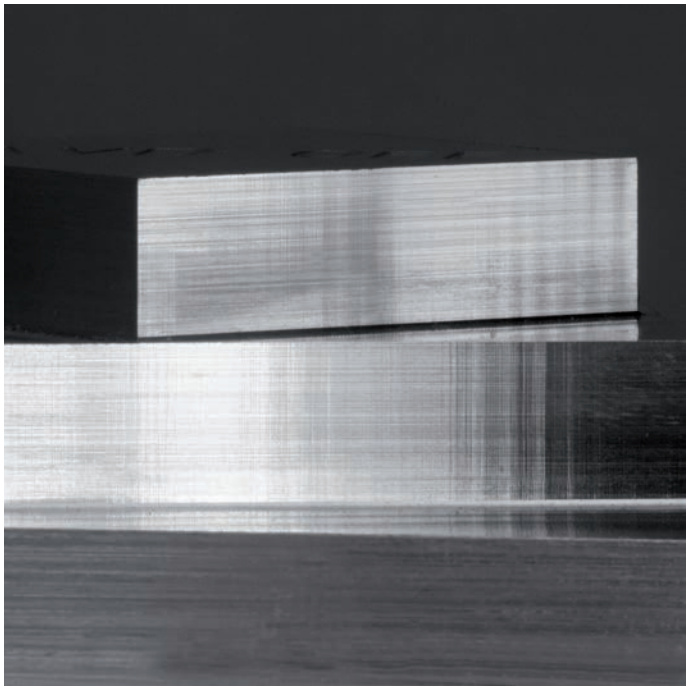
The hypernym **Dynamic Precision** stands for a number of HEIDENHAIN solutions for metal cutting that can dramatically improve the dynamic accuracy of a machine tool. It is the result of a new perspective on the competing demands for accuracy, high surface quality, and short machining times. The dynamic accuracy of machine tools manifests itself in deviations at the tool center point (TCP). These deviations depend on kinetic quantities such as velocity and acceleration (jerk), and are caused, in part, by vibrations of machine components.



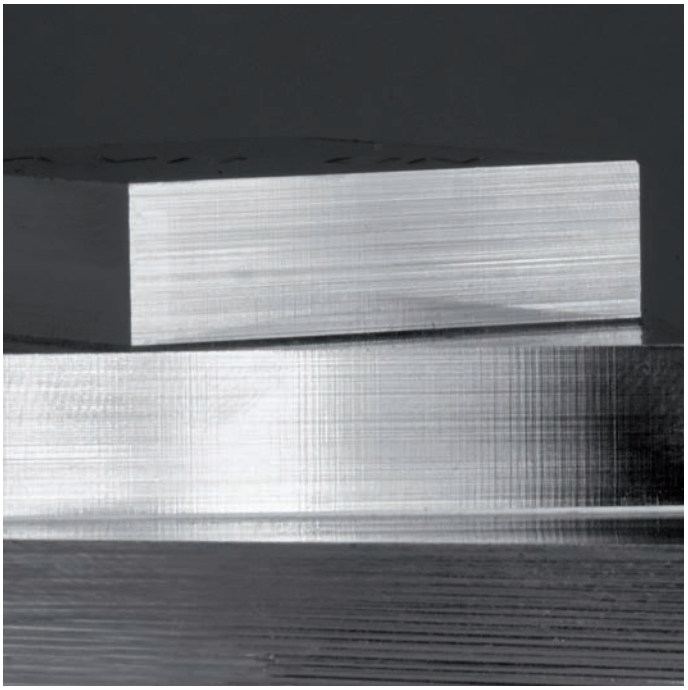
All these influences are together responsible for dimensional inaccuracies and faults in the workpiece surface. They therefore have a decisive impact on quality and, in the event of quality-related scrap, on productivity as well. Dynamic Precision counteracts these problems with intelligent control technology to enable designers to further improve the quality and dynamic performance of machine tools. This saves time and money in production.

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

- **CTC** – compensation of position errors due to machine elasticity between the encoder and the TCP. This increases accuracy during acceleration phases
- **AVD** – active vibration damping improves surfaces
- **PAC** – position-dependent adaptation of control parameters
- **LAC** – load-dependent adaptation of control parameters and maximum axis acceleration
- **MAC** – motion-dependent adaptation of control parameters



Vibrations can significantly impair surface quality



With AVD, visibly superior surface quality is achieved

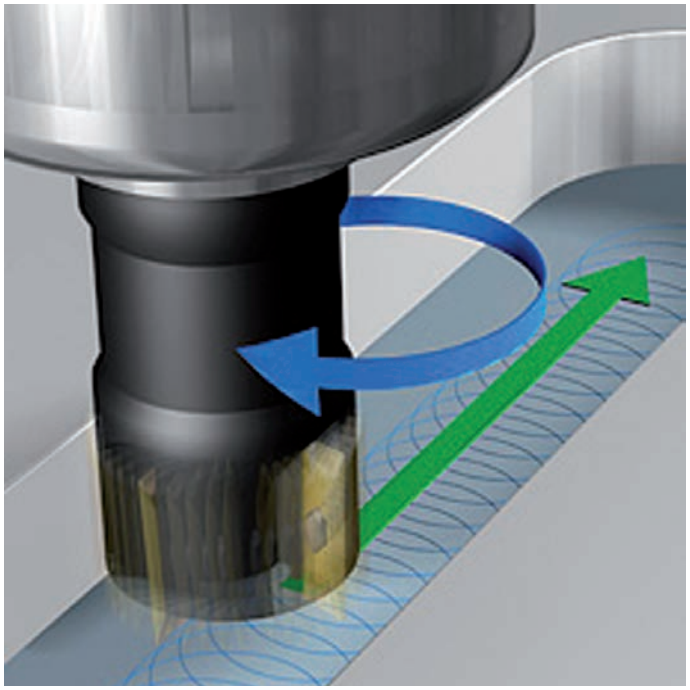
Quick and reliable machining with high contour fidelity
– Machining any contour slots with trochoidal milling

The benefit of trochoidal milling is its ultra-efficient machining of slots of all kinds. In this cycle, roughing is performed with circular movements, onto which a forward linear movement is superimposed. 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 hob'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.

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.

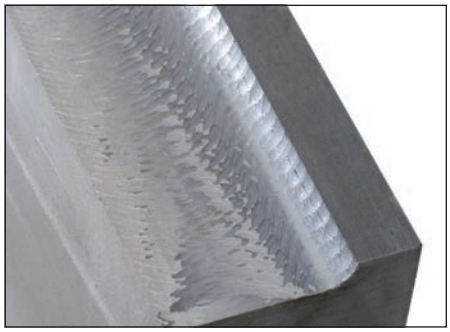
The benefits include:
- Entire length of cutter remains in contact with the workpiece
- Higher chip volume
- Relief from mechanical load on the machine
- Less vibration
- Integrated finishing of the side wall

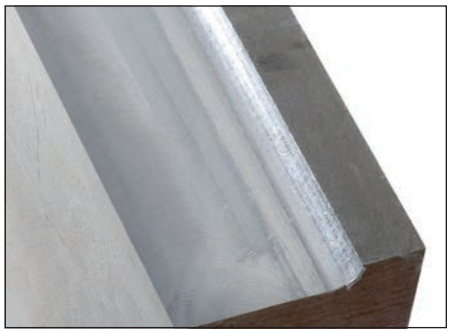


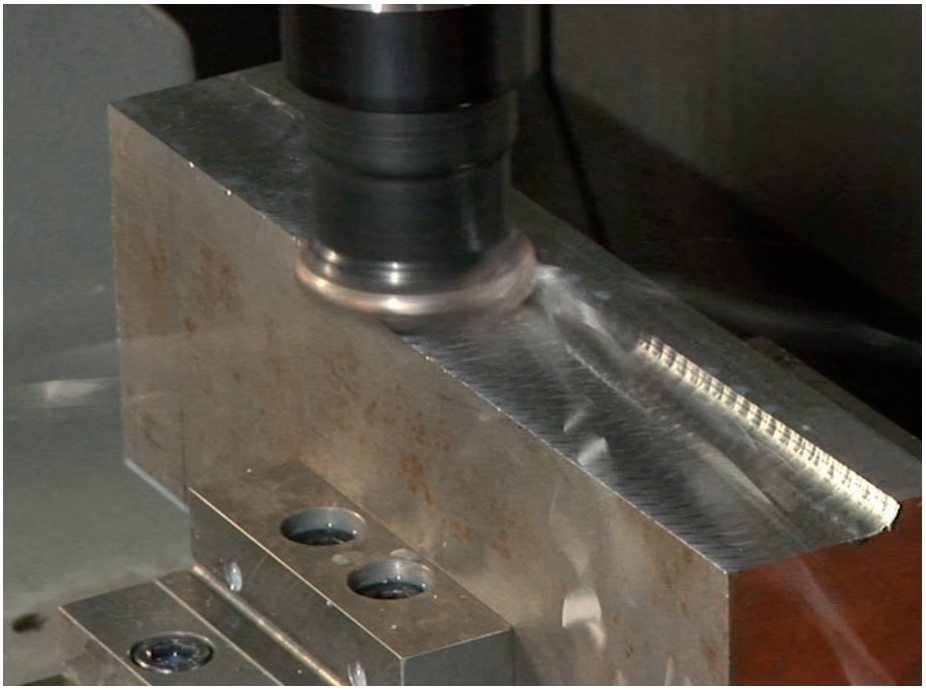
– Active Chatter Control option (ACC)

In high-performance milling, roughing operations give rise to strong milling forces. 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." Chatter subjects the machine to heavy strain and causes ugly marks on the workpiece surface. The tool, too, undergoes heavy and irregular wear due to chatter, even breaking in extreme cases.

With Active Chatter Control (ACC), HEIDENHAIN offers an effective control function for reducing a machine's tendency to chatter. The use of this control function is particularly advantageous during heavy machining. ACC makes substantially higher metal removal rates possible. This enables you to increase your metal removal rate by up to 25 % and more, depending on the type of machine. Thus, you can reduce the load on your machine while simultaneously increasing the life of your tools.







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

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

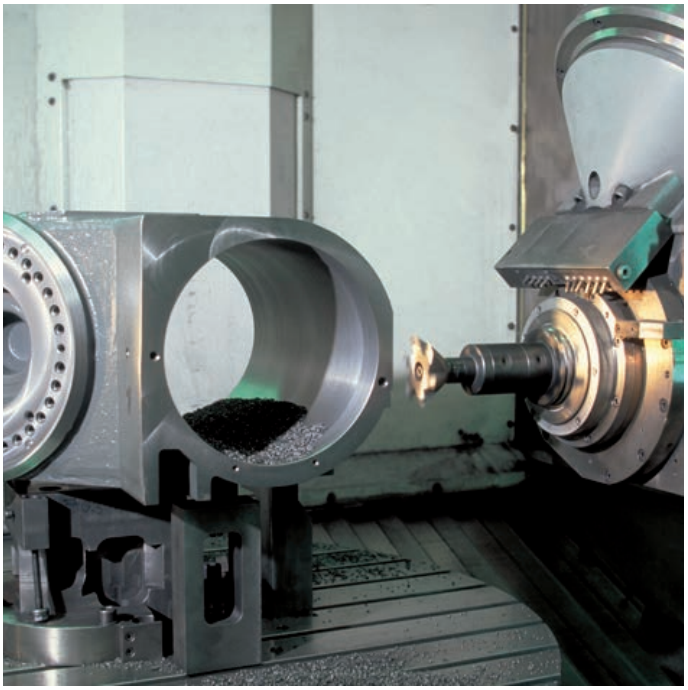
Many 5-axis operations that at first glance may seem very complex can be reduced to conventional 2-D movements that are simply tilted about one or more rotary axes or wrapped onto a cylindrical surface. The TNC supports you with application-oriented functions to help you write and edit such programs quickly and simply without a CAD/CAM system.

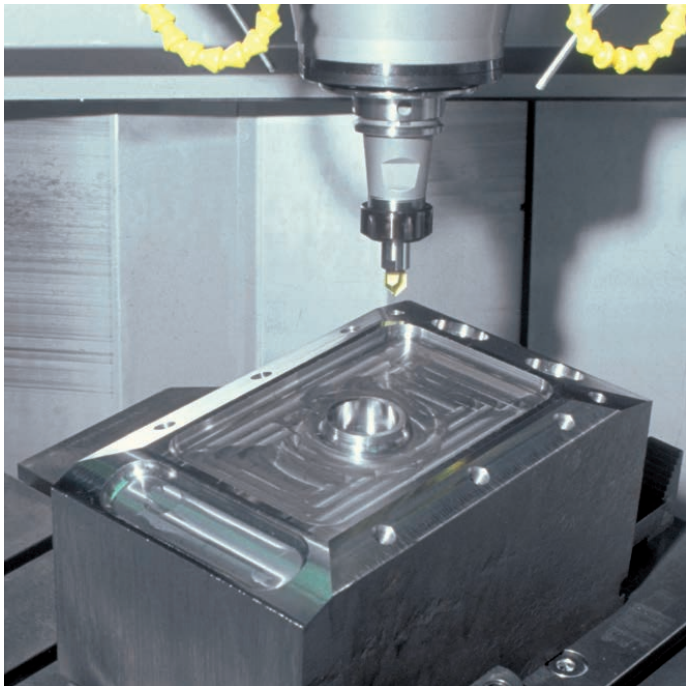
Tilting the working plane* (option 8)
Programs for contours and holes on inclined surfaces are often very complex and require time-consuming computing and programming work. Here the TNC 620 helps you to save a great deal of programming time.

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

The PLANE feature makes it easy to define a tilted working plane: you can specify tilted working planes in seven different ways, depending on the information on the workpiece drawing. 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.





Cylinder surface machining* (option 8)
With the TNC 620 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 620 then calculates and machines the corresponding cylindrical contour.

The TNC 620 features four cycles for cylindrical surface machining:
- Slot milling (the slot width is the same as the tool diameter)
- Guide-groove milling (the slot width is greater than the tool diameter)
- Ridge milling
- Outside contour milling

Manual axis motion in the tool direction on 5-axis machines
The safe retraction of a tool is very important with five-axis machining. The virtual tool axis function is of assistance here. You can use it to traverse the tool in the current direction of the tool axis 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 during machining with the handwheel in the active tool axis direction (option 21 required).

Feed rate for rotary tables in mm/min* (option 8)
By default, the feed rate of rotary axes is programmed in degrees/min. However, the TNC 620 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.

* The machine must be adapted to this function by the machine tool builder.

Minimize setup times
– The TNC 620 makes setup easy

Before you can begin machining, you must first clamp the workpiece and set up the machine, find the position and orientation of the workpiece on the machine, and set the workpiece preset. 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.

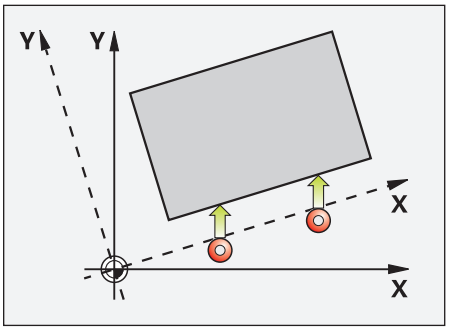
The TNC 620 features application-oriented, real-world setup functions. They support the operator, help to reduce non-productive time, and make overnight, unattended production possible. Together with the **touch probes**, the TNC 620 offers numerous probing cycles for automatic alignment of the workpieces, presetting, and measurement of the workpiece and the tool.

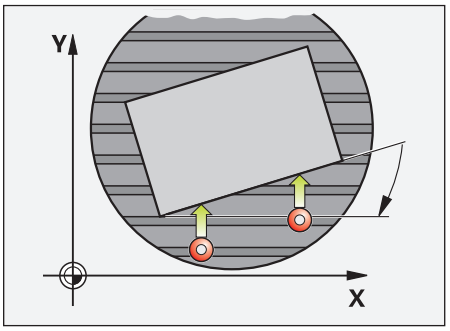
Delicate manual traverse
For setup, you can use the axis-direction keys to move the machine axes manually or in incremental jog. A simpler and more reliable way, however, is to use the electronic handwheels from HEIDENHAIN (see page 35). With the handwheels you are always close to the action, enjoy a close-up view of the setup process, and can control the infeed responsively and precisely.

Adapting the probing velocity
Frequently, the workpiece has to be probed at hidden locations or in cramped spaces. In this case, the standard probing feed rate is usually too fast. In such situations you can use the override knob to change the feed rate during probing. What is special about this option is that it does not influence accuracy.

Workpiece alignment (option 17)
The touch probes from HEIDENHAIN (see page 36) and the probing functions of the TNC 620 relieve you from the time-consuming alignment of the workpiece:
- Clamp the workpiece in any position.
- The touch probe determines the actual workpiece position by probing a surface, two holes, or two studs.
- The TNC 620 compensates the misalignment with a "basic rotation," which means that in the NC program the part is rotated by the measured misalignment, or the rotary table itself is turned to correct the misalignment.
- The TNC 620 offers manual, automatic, and semiautomatic cycles for correcting misalignments in two or three dimensions.

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





Setting presets
You can use a preset to assign a defined value in the TNC display to any workpiece position. Fast and reliable definition of the preset reduces nonproductive time and increases machining accuracy.

The TNC 620 features probing cycles (option 17) for automatic setting of presets. Once found, you can save these presets
- in the preset manager,
- in a datum table, or
- by directly setting the displayed value.

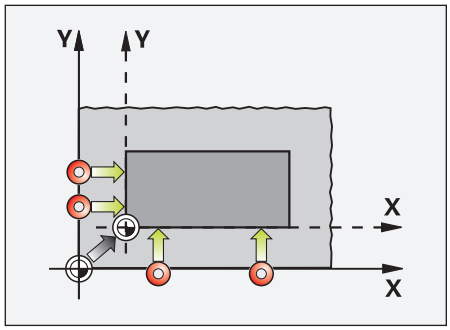
Preset management with the preset table
The preset manager makes flexible machining, shorter setup times, and increased productivity possible. In other words, it makes it much easier to set up the machine.

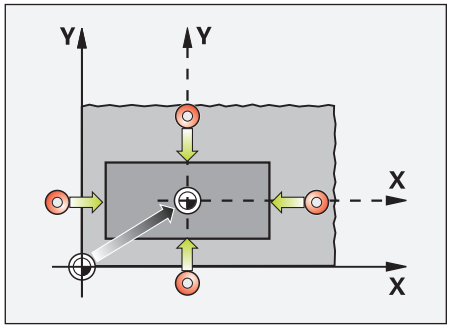
In the preset manager you can save **any number of presets** and assign an individual basic rotation to each one. In order to permanently save fixed presets in the machine working space, you can also write-protect individual lines.

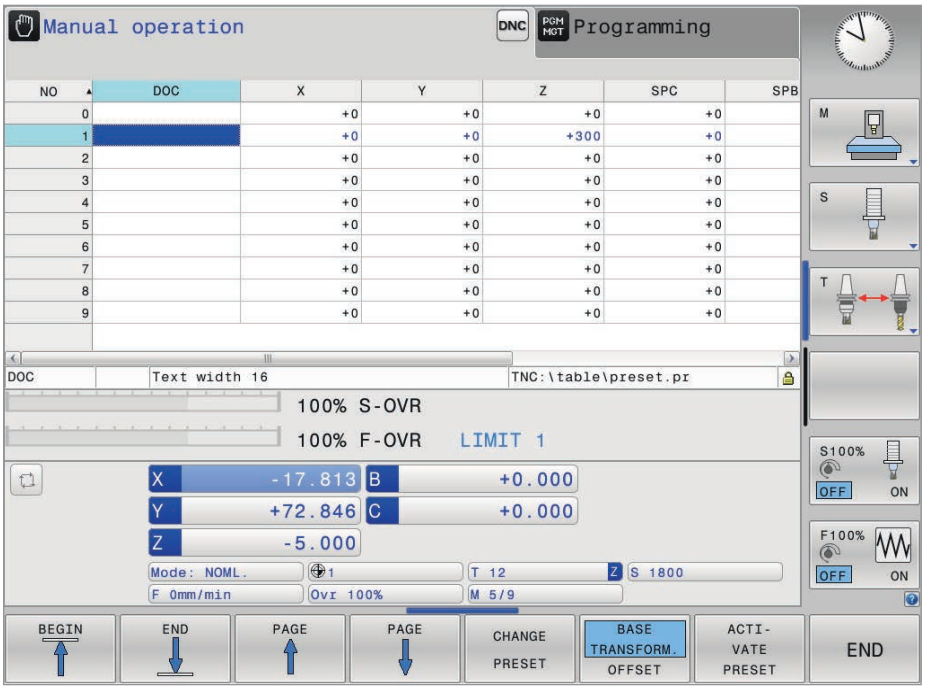
There are three ways to save presets rapidly in the preset manager:
- In the Manual mode by soft key
- By using the probing functions
- With the automatic probing cycles

Saving datums
In datum tables, you can save positions or values given or measured with respect to the workpiece. Datums are always relative to the active preset.

Setting a preset
at a corner, for example, or in the center of a circular stud







Automated machining
– The TNC 620 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 620 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.

Tool management
For machining centers with automatic tool changers, the TNC 620 offers a central tool memory for any number of tools. The tool memory is a freely configurable file and can therefore be optimally fitted to your needs. You can even have the TNC 620 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.

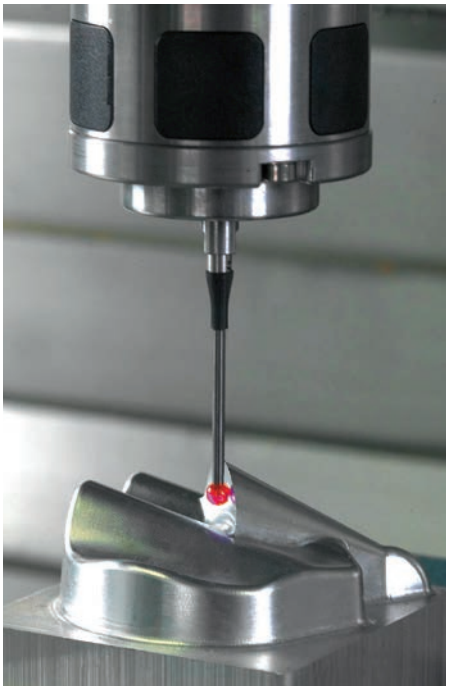
With the optionally available Extended Tool Management you can also graphically prepare and display any data.*

* The machine must be adapted to this function by the machine tool builder.

Inspecting workpieces for proper machining and dimensional accuracy (option 17)
The TNC 620 features a number of measuring cycles for checking the geometry of the machined workpieces. This requires the insertion of a 3-D touch probe from HEIDENHAIN (see page 36) instead of the tool in the spindle. 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.

Tool measurement and automatic compensation of tool data (option 17)
Together with the TT tool touch probe (see page 37), the TNC 620 offers the possibility of automatically measuring tools in the machine. The TNC 620 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 specified tolerances or if the monitored tool life has been exceeded, the TNC 620 locks the tool and automatically inserts a replacement tool.



– Pallet management and multiple machining

Pallet Management (option 22)
With pallet management, you can machine workpieces in any order automatically. When inserting the pallet, the associated machining program and the workpiece preset are automatically selected. Of course, you can also use coordinate transformations and measuring cycles in the machining programs.

Batch Process Manager (option 154)
Batch Process Manager is a powerful function for pallet machining and series production. With the clear-cut user interface, you plan your production process and receive important information about the upcoming operations.

Batch Process Manager automatically checks for missing tools, insufficient tool life, and required manual tool changes. The result of the check is displayed in the status overview.

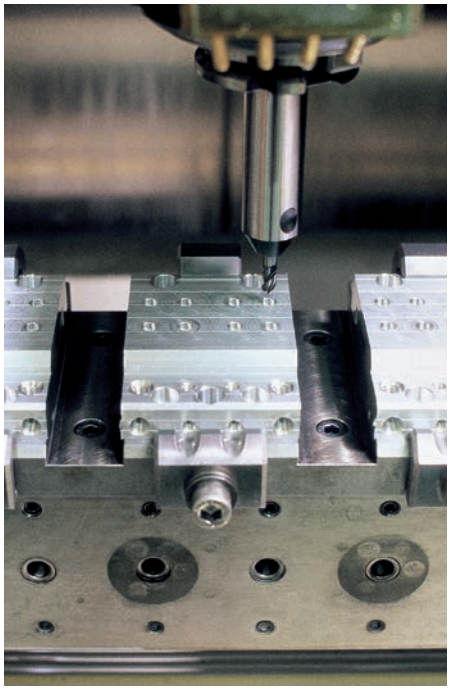
In Batch Process Manager, the following information is already displayed in advance:

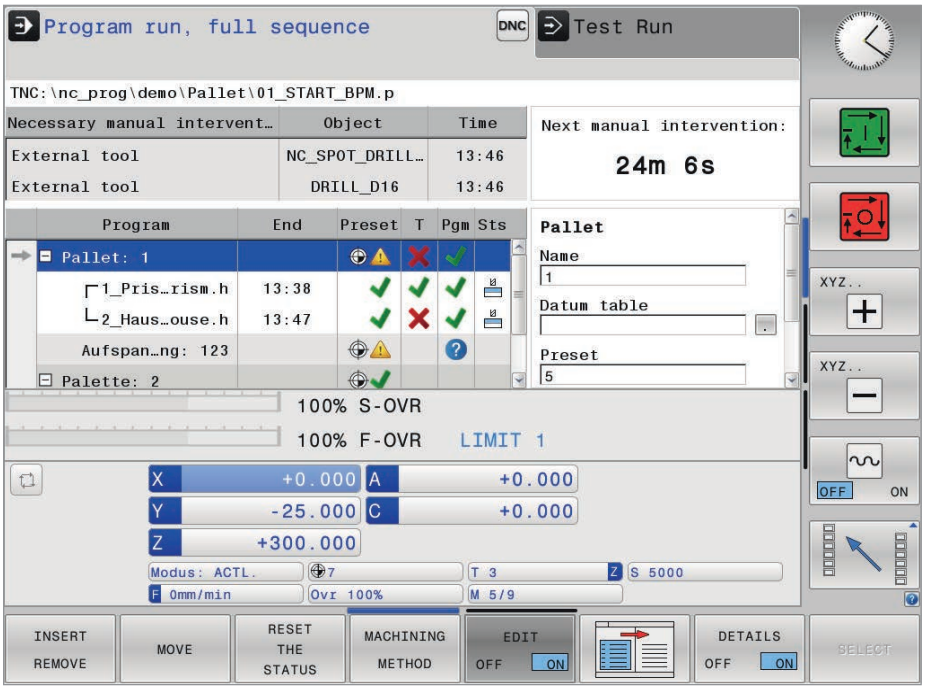
- Sequence of operations
- Time of next manual intervention
- Program duration and run time
- Status information regarding the preset, tool, and program

Tool-oriented machining (option 22)
In tool-oriented machining, one machining step is performed on all workpieces of a pallet before the next machining step. This reduces the number of tool changes to a necessary minimum and the machining time is significantly shorter.

The TNC 620 supports you with convenient fillable forms that allow you to assign a tool-oriented operation to a pallet with multiple workpieces. You can write the program, however, in the familiar workpiece-oriented sequence.

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





Programming, editing, testing
– The TNC 620 opens endless possibilities

The TNC 620 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 620 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 Klartext 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. You initiate a HEIDENHAIN Klartext dialog with a keystroke and the TNC immediately begins to support you actively in your work. Unambiguous questions and prompts help you enter all the required information.

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

Creating programs offline
The TNC 620 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. The TNC 620 can also run programs that were written in DIN/ISO format.

– Graphic support in any situation

Programming graphics
The two-dimensional programming graphics give you additional security: while you are programming, the TNC 620 draws every entered traverse command on the screen. You can select among the plan view, side view, and front view. Also, tool paths and rapid-traverse movements can be hidden, and the view can be scaled.

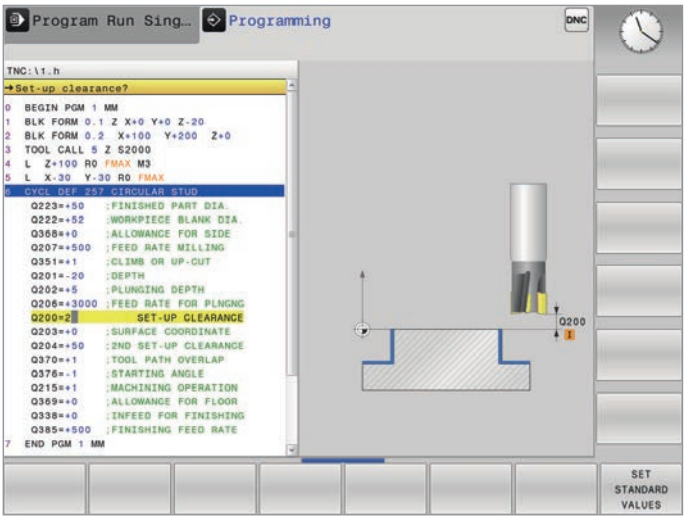
Program verification graphics (option 20)
To play it safe before running a program, the TNC 620 can simulate workpiece machining and display it with high graphic resolution. The TNC 620 can display the simulation in the following ways:

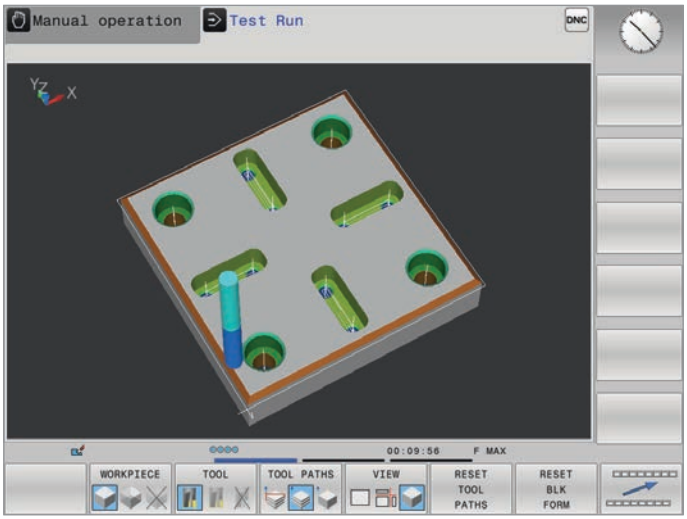
- In plan view with different shades of depth
- In three projections
- In 3-D view

You can adjust the type and quality of the image. Details can be displayed with magnification. In addition, the TNC 620 indicates the calculated machining time in hours, minutes, and seconds.

In the 3-D view, you can display the programmed tool-center 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. The TNC also features a measuring function in the 3-D view. You can position the mouse pointer anywhere in the graphic to see the coordinates.

Program-run graphics (option 20)
The program-run graphics display the workpiece in real time to show you the current stage of machining. Direct observation of the workpiece is usually impossible due to coolant and the safety enclosure. During workpiece machining you can switch between various operating modes at any time, for example to create programs. You then use free moments for a simple keystroke to take a glance at the progress of workpiece machining.





Programming in the workshop
– Straightforward function keys for complex contours

Programming 2-D contours

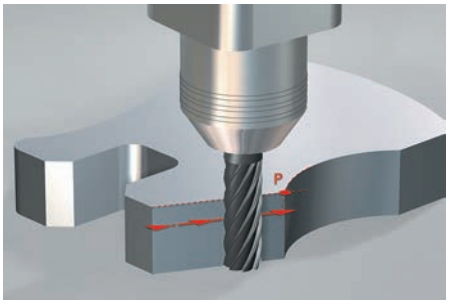
Two-dimensional contours are the bread and butter of a modern machine shop. Here the TNC 620 offers a variety of possibilities.

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

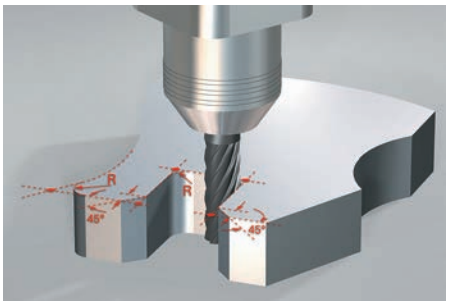
Straight and circular contour elements
To program a line segment, for example, simply press the key for linear traverse. The TNC 620 asks in Klartext format for all information required for a complete programming block, such as target coordinates, feed rate, tool compensation, and machine functions. Appropriate path function keys for circular movement, chamfers, and corner rounding simplify your programming. To avoid surface blemishes during approach to or departure from the contour, it must be approached smoothly—that is, tangentially.

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

The TNC 620 can look ahead over a radius-compensated contour for up to 99 blocks (option 21) to watch for back cutting and avoid contour damage such as can occur when roughing a contour with a large tool.

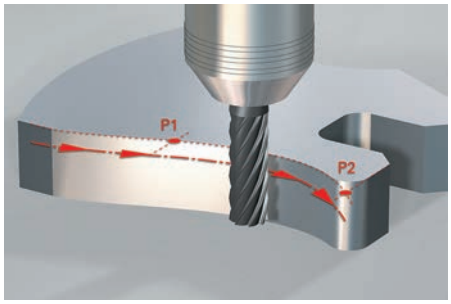


L
Straight line defined by its end point

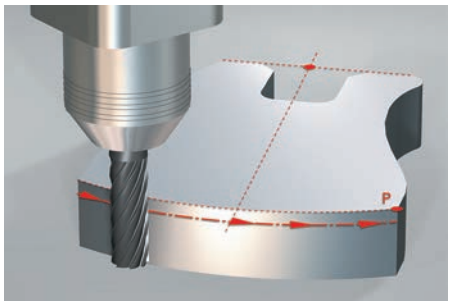


RND Corner rounding: circular path with smooth (tangential) connection on both sides, defined by radius and corner point

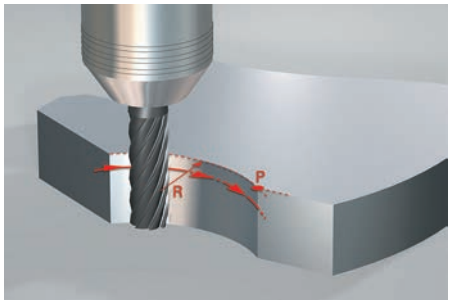
CHF Chamfer defined by the corner point and chamfer length



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



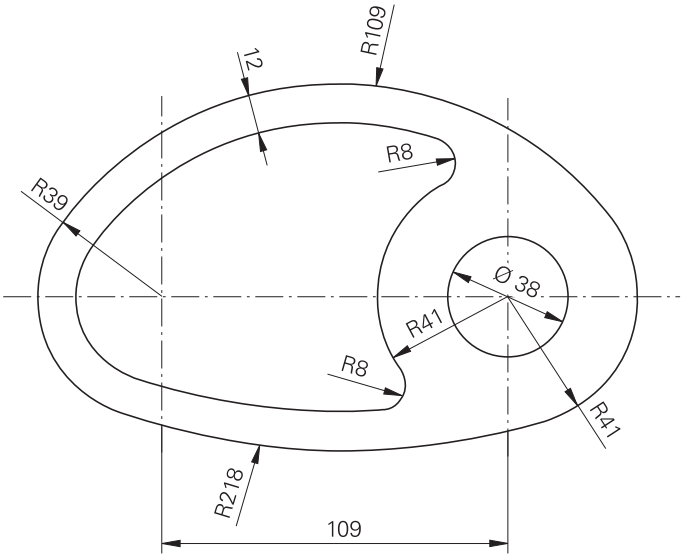
CC C
Circular path defined by its center, end point, and rotational direction

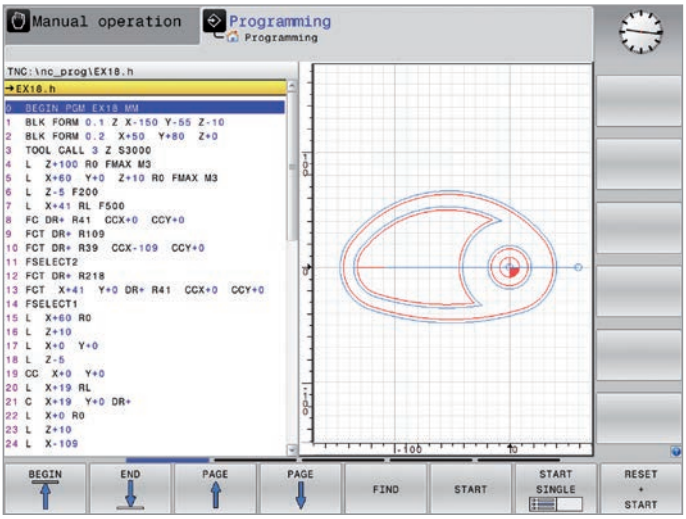


CR
Circular path defined by its radius, end point, and rotational direction

– Programming contours unconventionally

FK free contour programming (option 19)
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 it! It does not matter if individual contour elements are not completely defined as long as the complete contour has been. If the given data results in more than one mathematical solution, the helpful TNC 620 programming graphics show you the possible variants for your selection.





Programming in the workshop
– Field-proven cycles for recurring operations

Comprehensive fixed cycles for milling, drilling, and boring

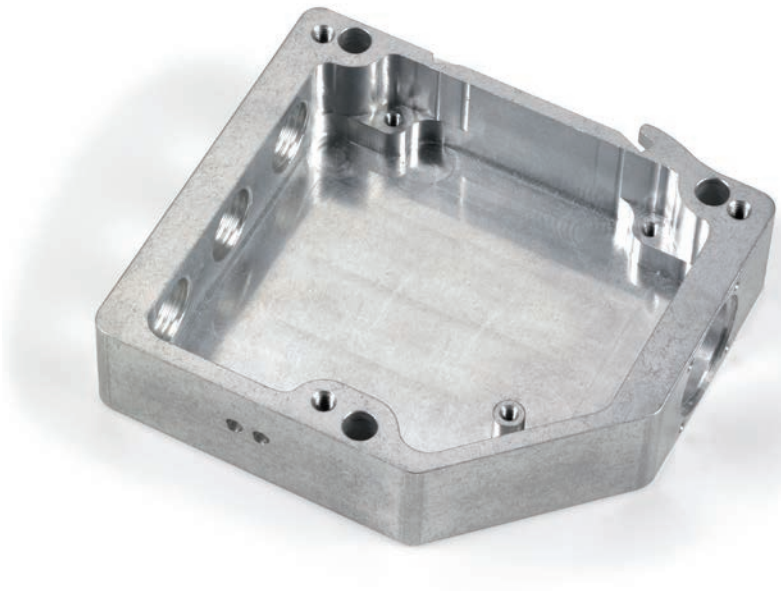
Frequently recurring operations that comprise several working steps are stored in the TNC 620 memory as standard cycles. You program them under conversational guidance and are supported by graphics that clearly illustrate the required input parameters.

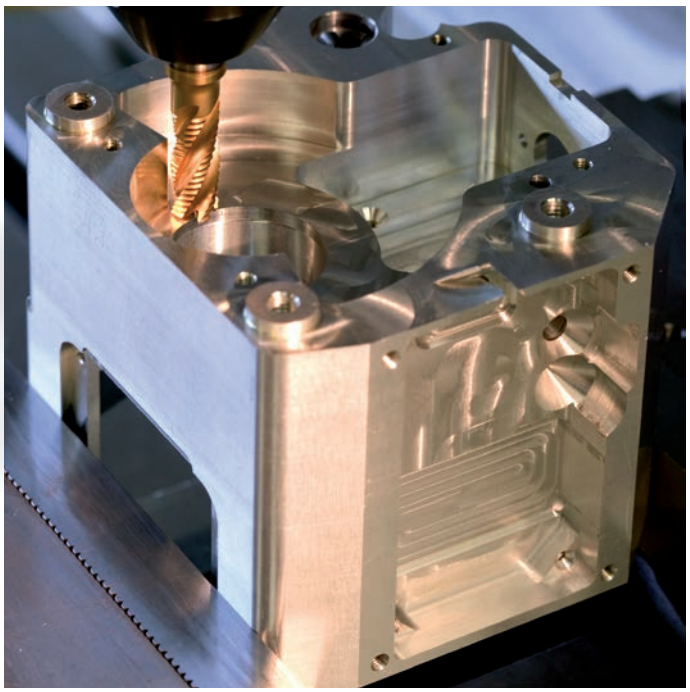
Standard cycles
In addition to the fixed cycles for drilling and tapping (with or without floating tap holder), optional cycles (option 19) are available for thread milling, reaming, engraving, and boring, as well as drilling patterns, milling cycles for face milling flat surfaces, for clearing and finishing pockets, slots, and studs.

Cycles for complex contours (option 19)
The **Subcontour List cycles** (SL) are particularly helpful for clearing pockets with combined contours. This term is used to identify machining cycles for pilot drilling, roughing, and finishing when the contour or subcontours are specified in subprograms. 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 620 maintains a **finishing allowance** on the wall and floor surfaces during roughing. When **roughing** with different tools, the control identifies material remaining in inside corners so that it can be cleared later with smaller tools. A separate cycle is used for milling to the finished dimension.

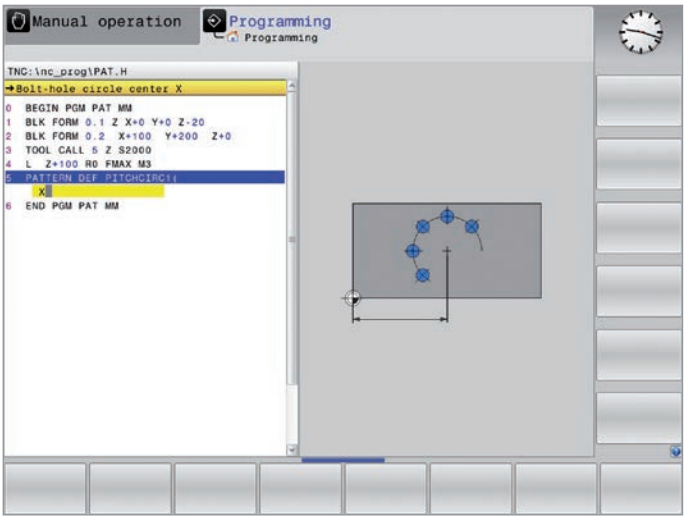




OEM cycles (option 19)
As original equipment manufacturers (OEMs), machine-tool builders can contribute their special manufacturing know-how by designing additional fixed cycles and saving them in the TNC 620. However, the end user can write his own cycles as well. HEIDENHAIN makes this possible with its PC program CycleDesign. This enables you to organize the input parameters and soft-key structure of the TNC 620 to suit your own needs.

Stay simple and flexible when programming machining patterns
Machining positions are often arranged in patterns on the workpiece. With the TNC 620, 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. Then you can machine all points at once or each point individually.

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
– 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 620 offers you a simple solution: coordinate transformation.

With coordinate transformation you can, for example, **rotate or mirror** the coordinate system, or **shift the datum**. With a **scaling factor** you can enlarge or reduce contours to respect shrinkage or oversizes.

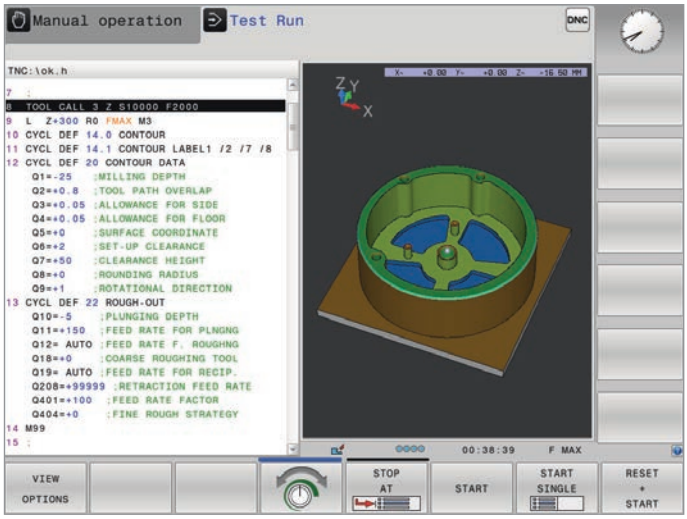
Program-section repeats and subprograms
Many machining operations repeat themselves either on the same workpiece or on different workpieces. Once you have programmed a detail, there is no reason to have to program it again. With its subprogramming feature, the TNC can save you a great deal of programming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

In **program-section repeats**, 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 about a programming step, but your User's Manual is not at hand? No problem: The TNC 620 numerical control and TNC 620 programming station now feature TNCguide, a convenient help system that can show the user documentation in a separate window.

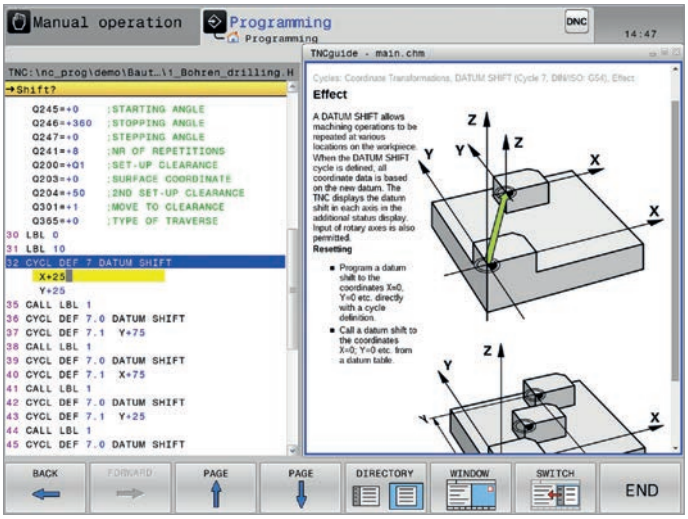
You can activate TNCguide by simply pressing the help key on the TNC keyboard or by clicking any soft key when the mouse pointer has a question mark. This can be done easily by clicking on the help icon displayed on the TNC screen.

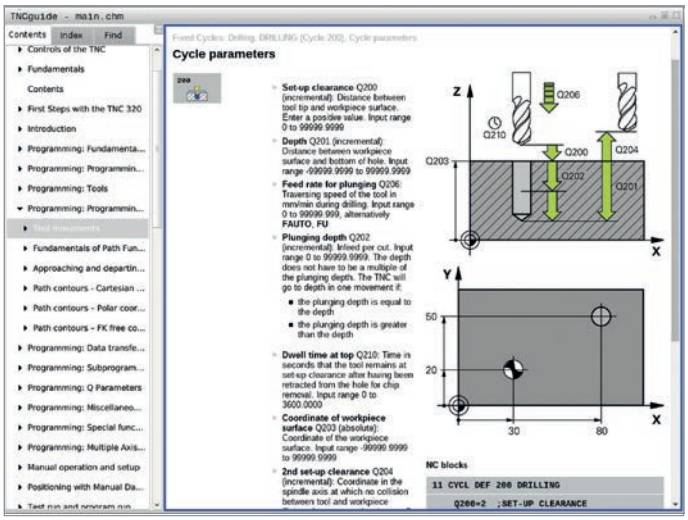
TNCguide usually displays the information in the immediate context of the element in question (context-sensitive help). This means that you immediately receive the relevant information. This function is particularly helpful with the soft keys. The method and effect of operation are explained in detail.

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 Klartext conversational programming
- Setup, Testing and Running NC Programs User's Manual
- User's manual for cycle programming
- User's manual for ISO programming
- User's manual for the TNC 620 programming station (installed only with the programming station)





Open for communication
– The TNC 620 understands CAD files

CAD viewer
The CAD viewer (standard feature) enables you to open 3-D CAD models and drawings directly on the TNC 620.

Various view options as well as rotation and zoom capabilities allow detailed visual control and analysis of your CAD data. Moreover, you can also use the viewer to find position values from the 3-D model. You simply select an arbitrary reference point in your drawing and select the desired contour elements. The CAD viewer then displays the coordinates of the elements in a window. The CAD viewer can depict the following file formats:

- STEP files (.STP and .STEP)
- IGES files (.IGS and .IGES)
- DXF files (.DXF)

CAD Import (option)
Why program complex contours if you already have the drawing in DXF, STEP, or IGES format? You can extract contours or machining positions from these CAD files. This not only saves time otherwise spent on programming and testing, you can also be sure that the finished contour is exactly according to the design engineer's specifications.

Extracting machining information directly from CAD data offers additional possibilities, in particular for creating NC programs with a tilted machining plane. You can also define the preset with a 3-D basic rotation of the 3-D model. Plus, you can place a datum with the appropriate 3-D rotation on the desired working plane.

You can easily save the working plane to the clipboard and transfer it to the NC program with the appropriate transformation and the associated PLANE command. On the defined working plane you can extract contours and machining positions and apply them to the NC program.

Selecting the contour is particularly convenient. 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 automatic contour detection. The TNC automatically selects all clearly identifiable contour elements until the contour closes or branches out. By this means, you can define extensive contours with just a few mouse clicks. Then you can simply copy the selected contour to an existing Klartext program via the clipboard.

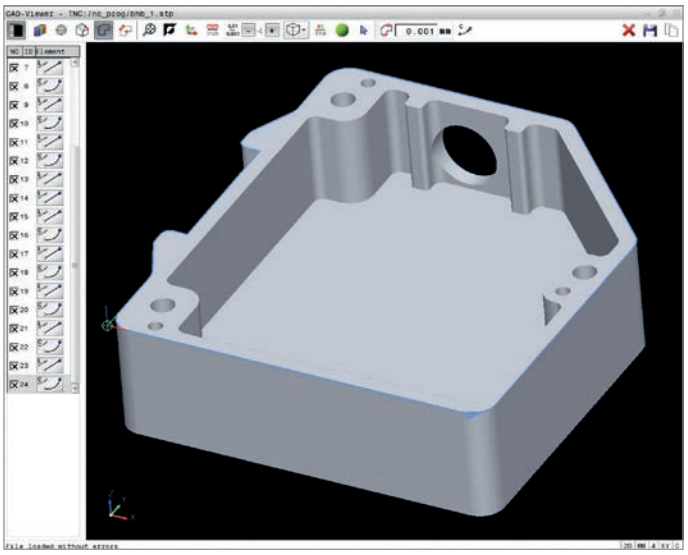
But you can also select **machining positions** and save them as point files, particularly in order to use drilling positions or starting points for pocket machining. This can be done very easily: using the mouse, simply select the desired area. In a pop-up window with a filter function, the TNC displays 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 CAD Import.

In addition, you can define the resolution of the contour program to be output in case you want to use it on older TNC controls, or a transition tolerance if occasionally the elements do not quite adjoin.

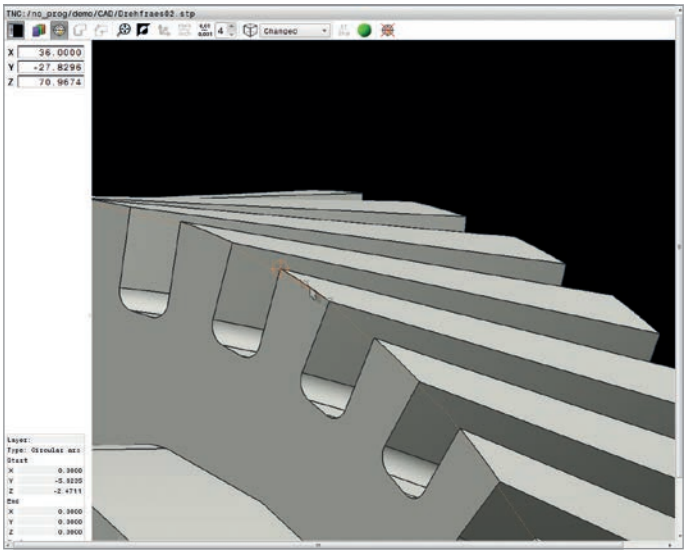
You can define the following locations as presets:

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

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



Contour selection from an imported CAD file



Display of a 3-D model in the CAD viewer

Open for communication
– Uniformly digital job management with Connected Machining

connected + machining

The well-functioning transfer of knowledge contributes decisively to the success of any company. To transfer knowledge quickly and without loss, effective communication via e-mail is just as much a matter of course as the universal availability of electronic production documents or the transfer of data to inventory management systems and production activity control systems. The tools and raw materials in stock, tool data, fixture setups, CAD data, NC programs, and inspection instructions must be available to machine operators across all shifts. Economic manufacturing therefore demands an efficiently working process chain and a networked control.

The TNC 620, with its **Connected Machining** package of functions, integrates itself flexibly into your process chain and helps you to optimize the transfer of knowledge within your company.

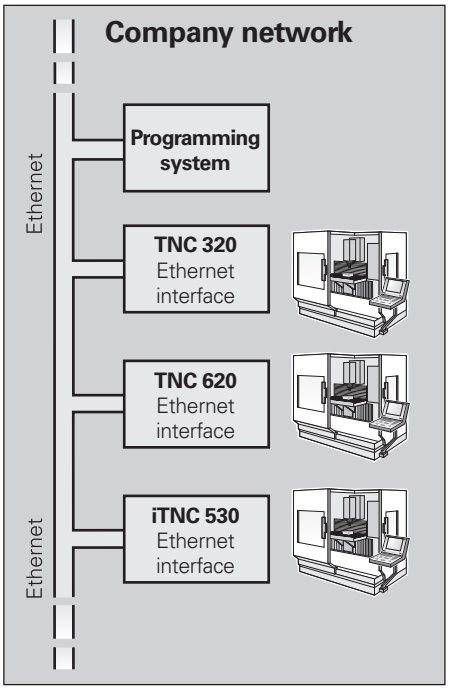
So let your workshop, as well, profit from all of the information available in your company. **Connected Machining** makes uniformly digital job management possible in networked manufacturing. You thus profit from:

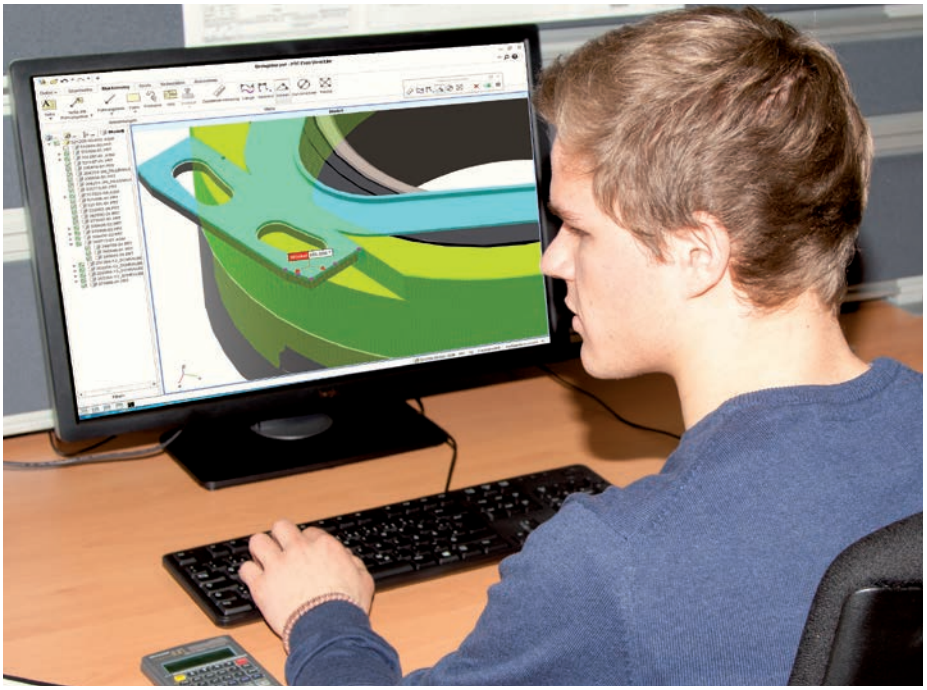
- Ease of data usage
- Time-saving procedures
- Transparent processes

The networked TNC 620
By integrating the TNC 620 with its **Connected Machining** functions in your company network, the control connects the workshop with PCs, programming stations, and other data storage devices in other areas of your company:

- Design
- Programming
- Simulation
- Production planning
- Production

Even in its standard version, the TNC 620 features a latest-generation Gigabit Ethernet interface in addition to its RS-232-C/V.24 data interface. The TNC 620 communicates with NFS servers and Windows networks in TCP/IP protocol without requiring additional software. The fast data transfer at rates of up to 800 Mbps guarantees very short transfer times. The TNC 620 therefore meets the ideal technological requirements for **Connected Machining,** the networking of the control in the workshop with all product-related areas in your company.





Standard performance range
Even the standard features of the TNC 620 offer many interesting applications. A CAD viewer, PDF viewer or the web browser Mozilla Firefox make the simplest form of **Connected Machining** possible: access to manufacturing process data right at the control.

The operation of web-based documentation software or ERP systems is just as much possible in this case as is the access to your e-mail inbox. The following file formats can also be opened directly on the TNC:
- Text files ending with .txt or .ini
- Graphic files ending with .gif, .bmp, .jpg, or .png
- Table files ending with .xls or .csv
- HTML files

Data transfer with Connected Machining
An additional solution for uniformly digital job management as part of **Connected Machining** is the free **TNCremo** software for PCs. With it, and even over the Ethernet interface, you can
- transfer remotely stored part programs and tool or pallet tables in both directions, and
- start the machine.

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

Provision of operating states
Option 137, the State Reporting Interface (SRI), provides you with a simple and reliable interface for recording the operating data of your machine. It records the idle times, run times, and fault messages of your machine. SRI also provides historical operating data, which can even be recalled if the company network was down for several hours.

Using job-related data on the control
With the **REMOTE DESKTOP MANAGER** (option 133) you operate a Windows PC directly from the TNC 620. You can access EDP systems that are part of the process chain directly from the control, and you can also benefit from considerably more efficient setup processes by eliminating burdensome legwork between the machine and the front office. Technical drawings, CAD data, NC programs, tool data, work instructions, parts lists, and warehouse information are digitally available at the machine. E-mails can be sent and received with ease. With a simple keystroke on the machine operating panel you can switch between the control screen and the screen of the Windows PC. This PC can be a computer in the local network or an industrial PC (IPC) in the machine's electrical cabinet.

Detailed data for the optimal organization of the production process
HEIDENHAIN DNC* enables, among other things, the connection of TNC controls to inventory management systems and production activity control systems. Automatic feedback messages about active production processes, for example, can be set up over this interface.

* The machine must be adapted to this function by the machine tool builder.

One aspect of Connected Machining is the **StateMonitor*** PC software, which gives you access to the status of the current machining operations. You can use the software with any device that has a web browser. This allows you to use StateMonitor not only on your control or PC but also on your smartphone or tablet. In a well-designed display, you can quickly gain an overview of the current machine status or see whether any machine messages are pending. This allows you to react immediately and take appropriate action. You can also easily configure StateMonitor to send you an e-mail for specific events, such as at the end of a program, for a machine stop, or for a servicing message.

* Option 18 required;
recommended as of NC SW 81760x-04 SP4

Open for communication
– The TNC 620 programming station

Why a programming station?
It is, of course, easy to write a part program with the TNC at the machine, even while the machine tool is machining a different part. Nevertheless, short reloading times and other machining tasks can often hinder any prolonged or concentrated programming work. With the TNC 620 programming station you have the capability of programming just as you do at the machine, but away from all the noise and distractions of the shop floor.

Creating programs
The programming, testing, and optimization of HEIDENHAIN Klartext or G-code programs with the programming station significantly reduce machine idle time. This does not require a shift in thinking, since every keystroke feels the same. This is because, on the programming station, you program on the same keyboard as the one on the machine.

Testing of programs created offline
Naturally, 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 620 programming station is based on the same software as the TNC 620, it is also ideally suited for training purposes. The program is entered on the original keyboard unit. Even the test run functions exactly as it does on the machine. This gives trainees the confidence that they will need for working on the actual machine.

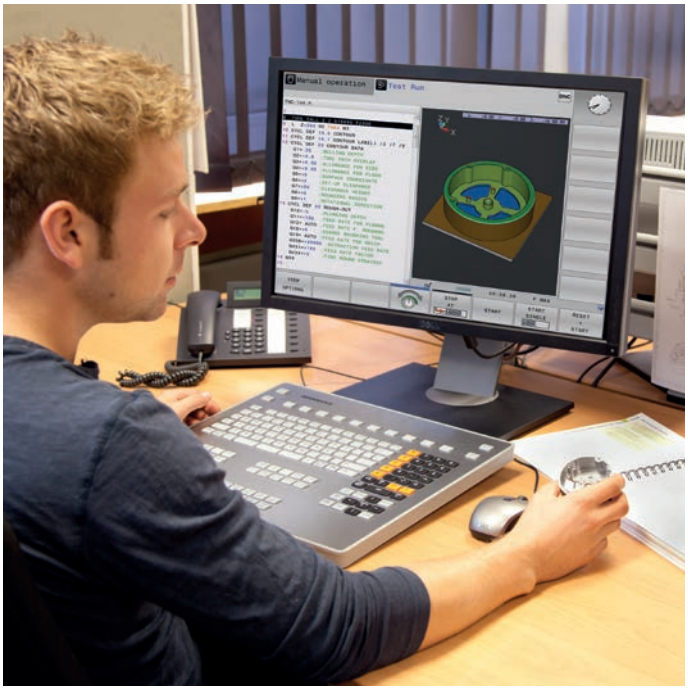
Because the TNC 620 can be programmed in Klartext and with G codes, the TNC 620 programming station can also be used in schools for TNC programming training.

Your workstation
The programming station software runs on a PC. The PC screen shows you the same 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** includes all functions of the TNC 620, and permits short programs to be saved. It is programmed via the PC keyboard.

On the version with a **TNC operating panel** you then create your programs as always, on a keyboard with the same function keys as on the control of 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 620's most important dialog initiation keys.



Programming station with TNC operating panel

Further information:
Comprehensive descriptions of the programming station and a free demo version are available on the Internet at *www.heidenhain.de*. Or simply ask for the *TNC Programming Station* CD or brochure.

Positioning with the electronic handwheel
– Delicate axis traverse

You can use the direction keys to move the machine axes manually while setting up the workpiece. A simpler and more sensitive way, however, is to use the electronic handwheels from HEIDENHAIN.

You can move the axis slide via the feed motors in direct relation to the rotation of the handwheel. For delicate operations, you can set the transmission ratio incrementally to a defined distance per handwheel revolution.

Panel-mounted handwheels
The HR 130 and HR 150 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 panel-mounted handwheels

Portable handwheels
The portable HR 510, HR 520, and HR 550 handwheels are particularly helpful for when you have to work close to the machine's working space. The axis keys and certain function 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. The HR 520 and HR 550 handwheels feature an integrated display for user-friendly remote operation of the control. As a wireless handwheel, the HR 550 is ideal for use on large machines. If you no longer need the handwheel, just attach it to the machine somewhere by its built-in magnets.

The following functions are available on the HR 520 and 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 potentiometers 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 to be defined by the machine tool builder

HR 550

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 620's cycles, 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 probes* are inserted directly into the machine tool spindle. They can be equipped with various shanks depending on the machine. The ball tips, which are made of ruby, are available in different diameters with various stylus lengths.

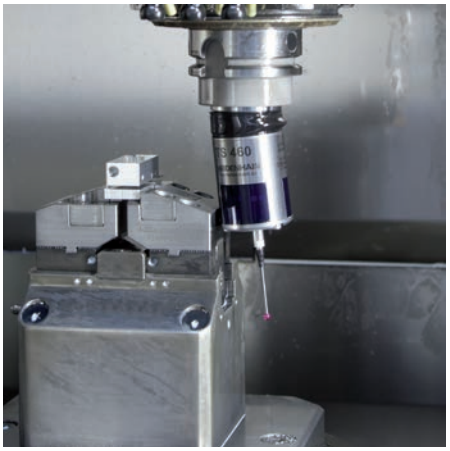
* Machine and TNC must be specially adapted by the machine tool builder.

Touch probes with **cable connection for signal transmission** for machines with manual tool changing, as well as for grinding machines and lathes:

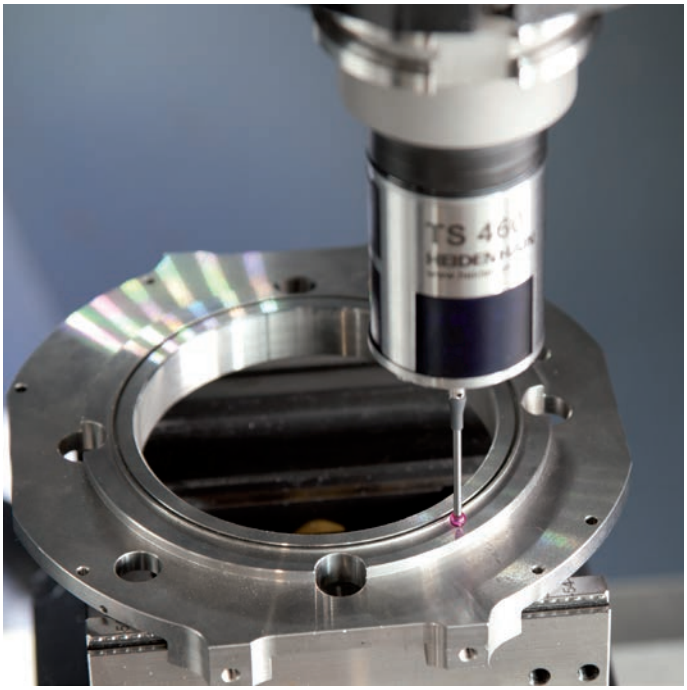
TS 260 – new generation, axial or radial cable connection

Touch probes with **radio or infrared signal transmission** for machines with an automatic tool changer:

TS 460 – new-generation standard touch probe for radio and infrared transmission with compact dimensions, energy-saving mode, optional collision protection, and thermal decoupling
TS 642 – activation by switch in the taper shank, infrared transmission
TS 740 – high probing accuracy and repeatability, low probing force, with infrared transmission



TS 460 with collision protection



TS 460

SE 660

Further information:

Detailed descriptions about workpiece touch probes are available on the Internet at *www.tastsysteme.de* or in the *Touch Probes* brochure.

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

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

These touch probes 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. During probing of the rotating or stationary tool, such as for individual tooth measurement, the contact plate is deflected, and a trigger signal is transmitted directly to the TNC 620.

The **TT 160** uses cable-bound signal transmission, whereas the **TT 460** sends its signals via radio or infrared transmission. It is thus particularly well-suited for use on rotary and tilting tables:

TT 460

Further information:

Detailed descriptions about tool touch probes are available on the Internet at *www.tastsysteme.de* or in the *Touch Probes* brochure.

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

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

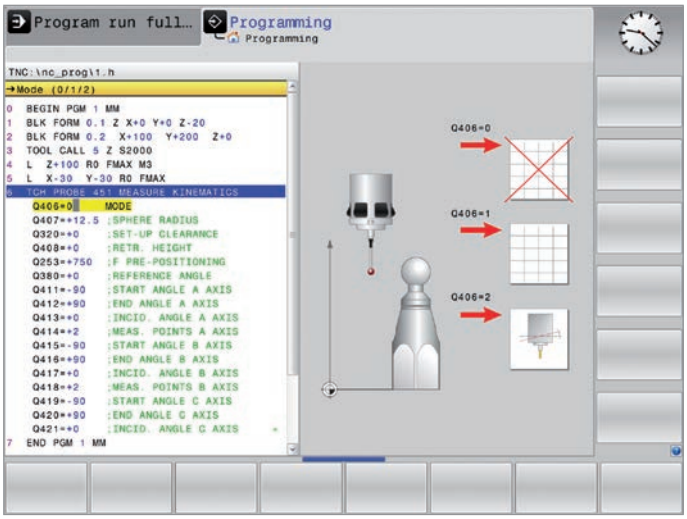
The TNC function **KinematicsOpt** is an important building block for meeting these demanding requirements. With an inserted HEIDENHAIN touch probe, a 3–D touch probe cycle measures your machine's rotary axes fully automatically. Measurement is the same, regardless of whether the axis is a rotary table, a tilting table, or a swivel head.

In order to measure the rotary axes, a calibration sphere is mounted 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 the range that you want to measure for each rotary axis.

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 containing the actual measured values and the measured and optimized dispersion (measure of 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. For this reason, HEIDENHAIN offers calibration spheres with highly rigid holders that are available in various lengths.



Overview
– Specifications

Specifications	Standard	Option	
Components	•		MC main computer with operating panel and integrated 15-inch or 19-inch screen (portrait orientation) for multitouch operation MC main computer with separate TE 730 or TE 735 operating panel and integrated 15-inch screen
Operating system	•		HEROS 5 real-time operating system for machine control
Memory	•		1.8 GB (on CFR compact flash memory card) for NC programs
Input resolution and display step	• •	 23 23	Linear axes: to 0.1 µm Rotary axes: to 0.0001° Linear axes: to 0.01 µm Rotary axes: to 0.000 01°
Input range	•		Maximum 999 999 999 mm or 999 999 999°
Interpolation	• • •	 9 8 	Linear in 4 axes Linear in 5 axes (requires an export license) Circular in 2 axes Circular in 3 axes with tilted working plane Helical: superimposition of circular and straight paths
Block-processing time	•		1.5 ms (3-D straight line without radius compensation)
Axis feedback control	• •		Position loop resolution: Signal period of the position encoder/1024 Cycle time of interpolator: 3 ms
Range of traverse	•		Maximum 100 m
Spindle speed	•		Maximum 100 000 rpm (with 2 pole pairs)
Error compensation	• •		Linear and nonlinear axis error, backlash, reversal spikes during circular movements, thermal expansion Static friction, sliding friction, reversal error
Data interfaces	• • • •	 18	RS-232-C/V.24 max. 115 kbps Extended data interface with LSV-2 protocol for remote operation of the TNC 620 over the data interface with the HEIDENHAIN software TNCremo or TNCremoPlus Gigabit Ethernet interface 1000BASE-T 5 x USB (1 x front USB 2.0; 4 x back panel USB 3.0) 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

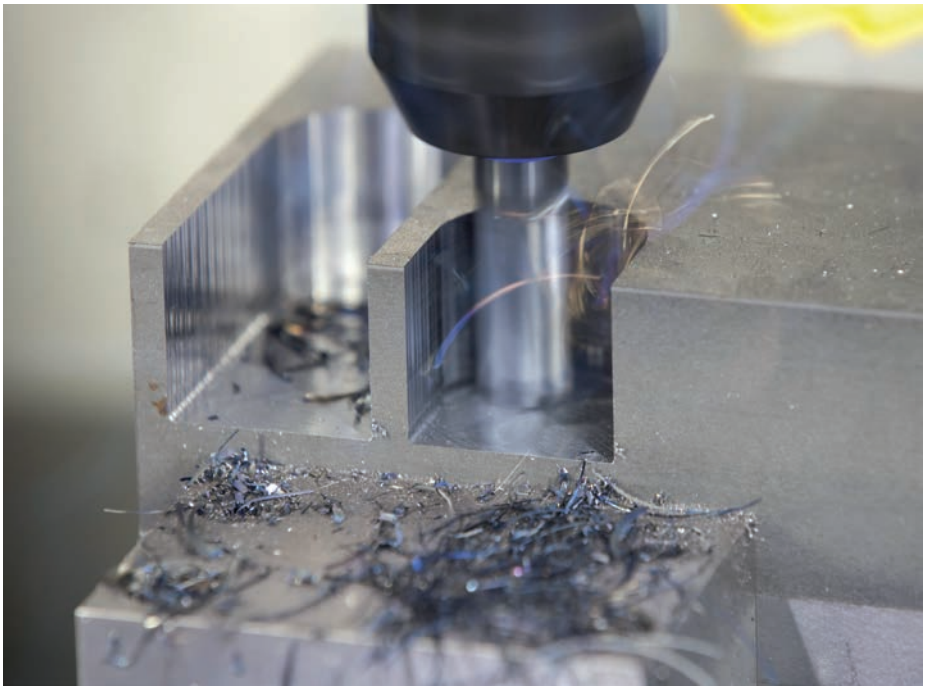
Overview
– User functions

User functions	Standard	Option	
Short description	• •	 0/1 	Basic version: 3 axes plus spindle One or two additional NC axes Digital current and shaft speed control
Program entry	• •	 42	In HEIDENHAIN Klartext conversational programming language ISO (*with integrated operating panel:* via soft keys or external standard USB keyboard; *with separate operating panel:* via ASCII keyboard) Direct loading of contours or machining positions from CAD files (STP, IGS, DXF) and saving as Klartext contouring program or as Klartext point table
Position entry	• • •		Nominal positions for straight lines and arcs in Cartesian coordinates or polar coordinates Incremental or absolute dimensions Display and entry in mm or inches
Tool compensation	•	 21 9	Tool radius in the working plane and tool length Radius-compensated contour look-ahead for up to 99 blocks (M120) Three-dimensional tool-radius compensation for the later changing of tool data without needing to recalculate the 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	•	 9 9 9 9	Motion control with highly smoothed jerk 3-D tool compensation through surface normal vectors Keeping the tool perpendicular to the contour Tool radius compensation perpendicular to the tool direction Manual traverse in the active tool-axis system
Rotary table machining		8 8	Programming of cylindrical contours as if in two axes Feed rate in distance per minute
Contour elements	• • • • • • •		Straight line Chamfer Circular path Circle center Circle radius Tangentially connecting circular arc Corner rounding
Approaching and departing the contour	• •		Via straight line: tangential or perpendicular Via circular arc
FK free contour programming		19	FK free contour programming in HEIDENHAIN Klartext format with graphic support for workpiece drawings not dimensioned for NC
Program jumps	• • •		Subprograms Program-section repeats Calling any program as a subprogram
Teach-In	•		Actual positions can be transferred directly into the NC program

User functions	Standard	Option	
Fixed cycles	•		Drilling, conventional and rigid tapping, rectangular and circular pockets
		19	Peck drilling, reaming, boring, counterboring, centering
		19	Milling of internal and external threads
		19	Clearing of level and oblique surfaces
		19	Multi-operation machining of rectangular and circular pockets, rectangular and circular studs
		19	Multi-operation machining of straight and circular slots
		19	Cartesian and polar point patterns
		19	Contour train, contour pocket
		19	Contour slot with trochoidal milling
		19	Engraving cycle: engraving of text or numbers in a straight line or on an arc
		19	OEM cycles (special cycles developed by the machine tool builder) can be integrated
Coordinate transformation	•		Shifting, rotating, mirroring, scaling (axis-specific)
		8	Tilting the working plane, PLANE function
Q parameters Programming with variables	•		Mathematical functions =, +, –, *, /, sin α, cos α, tan α, arc sin, arc cos, arc tan, a^n, e^n, ln, log, $\sqrt{a}$, $\sqrt{a^2 + b^2}$
	•		Logical operations (=, = /, <, >)
	•		Calculating with parentheses
	•		Absolute value of a number, constant π, negation, truncation of digits before or after the decimal point
	•		Functions for calculation of circles
	•		Functions for text processing
Programming aids	•		Calculator
	•		Complete list of all current error messages
	•		Context-sensitive help function for error messages
	•		TNCguide: the integrated help system. User information available directly on the TNC 620
	•		Graphic support for the programming of cycles
	•		Comment and structure blocks in the NC program
Program verification graphics Display modes		20	Graphic simulation before a program run, even while another program is running
		20	Plan view / projection in three planes / 3-D view, also in tilted working plane / 3-D line graphics
		20	Detail zoom
Programming graphics	•		In the Programming and Editing mode, the contours of the NC blocks are drawn on screen while they are being entered (2-D pencil-trace graphics), even while another program is running
Program-run graphics Display modes		20	Graphic simulation during real-time machining
		20	Plan view / view in three 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
Preset management	•		For saving any presets
Datum tables	•		Multiple datum tables for storing workpiece-specific datums
Pallet tables		22	Workpiece-oriented execution of pallet tables with any number of entries for selection of pallets, NC programs, and datums
		154	Planning the production process with Batch Process Manager

Overview
– User functions (continued)

User functions	Standard	Option	
Touch probe cycles		17 17 17 17	Touch probe calibration Compensation of workpiece misalignment, manual or automatic Presetting (manual or automatic) Automatic tool and workpiece measurement
Parallel secondary axes	• • •		Compensation of movement in the secondary axes U, V, W through the principal axes X, Y, Z Movements of a parallel axis included 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, Czech, French, Italian, Spanish, Portuguese, Swedish, Danish, Finnish, Dutch, Polish, Hungarian, Russian (Cyrillic), Chinese (traditional, simplified), Slovenian, Slovak, Norwegian, Korean, Turkish, Romanian
CAD viewer	•		Display of CAD models on the TNC



– Accessories

Accessories	
Electronic handwheels	• One **HR 510** portable handwheel, or • One **HR 520** portable handwheel with display, or • One **HR 550** portable wireless handwheel with display, or • One **HR 130** panel-mounted handwheel, or • Up to three **HR 150** panel-mounted handwheels via HRA 110 handwheel adapter
Workpiece measurement	• **TS 230** workpiece touch trigger probe with cable connection, or • **TS 440** workpiece touch trigger probe with infrared transmission, or • **TS 444** workpiece touch trigger probe with infrared transmission, or • **TS 640** workpiece touch trigger probe with infrared transmission, or • **TS 740** workpiece touch trigger probe with infrared transmission
Tool measurement	• **TT 160** tool touch trigger probe, or • **TS 460** tool touch trigger probe with infrared transmission
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 with virtual keyboard or 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** software for data transfer—free of charge • **TNCremoPlus** software for data transfer with live-screen function

Overview
– Options

Option number	Option	As of NC software 81776x-	ID	Comment
0	Additional Axis	01	354540-01	Additional control loop 1
1	Additional Axis	01	353904-01	Additional control loop 2
8	Advanced Function Set 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	Advanced Function Set 2	01	617921-01	**Interpolation:** linear in 5 axes (requires export license) **3-D machining** • 3-D tool compensation through surface normal vectors • Using the electronic handwheel to change the angle of the swivel head during program run without affecting the position of the tool point (TCPM = Tool Center Point Management); only with option 21 • Keeping the tool perpendicular to the contour • Tool radius compensation perpendicular to the tool direction • Manual traverse in the active tool-axis system
17	Touch Probe Functions	01	634063-01	**Touch probe cycles** • Compensation of workpiece misalignment, setting of presets • Automatic tool and workpiece measurement • Touch probe input enabled for non-HEIDENHAIN touch probes
18	HEIDENHAIN DNC	01	526451-01	Communication with external Windows applications over COM component
19	Advanced Programming Features	01	628252-01	**FK free contour programming** **Fixed cycles** • Peck drilling, reaming, boring, counterboring, centering • Milling of internal and external threads • Clearing of level and oblique surfaces • Multi-operation machining of straight and circular slots • Multi-operation machining of rectangular and circular pockets • Cartesian and polar point patterns • Contour train, contour pocket—also with contour-parallel machining • Contour slot with trochoidal milling • Special cycles developed by the machine tool builder can be integrated
20	Advanced Graphic Features	01	628253-01	**Program-verification graphics, program-run graphics** Plan view, projection in three planes, 3-D view, 3-D line graphic
21	Advanced Function Set 3	01	628254-01	**Tool compensation** • Radius-compensated contour look-ahead for up to 99 blocks (LOOK AHEAD) **3-D machining** • Superimposing handwheel positioning during program run

Option number	Option	As of NC software 81776x-	ID	Comment
22	Pallet Management	01	628255-01	**Pallet management**
23	Display Step	01	632986-01	**Display step** to 0.01 µm or 0.000 01°
24	Gantry Axes	01	634621-01	Gantry axes in master-slave torque control
42	CAD Import	01	526450-01	Importing of contours from 3-D models
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
49	Double Speed	01	632223-01	Short control-loop cycle times for direct drives
93	Extended Tool Management	01	676938-01	Extended tool management for more information
133	Remote Desk. Manager	01	894423-01	Display and remote operation of external computer units (e.g., a Windows PC)
137	State Reporting Interface	06	1232242-01	Provision of operating states
141	Cross Talk Comp.	01	800542-01	CTC: compensation of axis couplings
142	Pos. Adapt. Control	01	800544-01	PAC: position-dependent adaptation of the control parameters
143	Load Adapt. Control	01	800545-01	LAC: load-dependent adaptation of the control parameters
144	Motion Adapt. Control	01	800546-01	MAC: motion-dependent adaptation of control parameters
145	Active Chatter Control	01	800547-01	ACC: active suppression of chatter during heavy machining
146	Active Vibration Damping	01	800548-01	AVD: active suppression of vibration for better surfaces
154	Batch Process Manager	05	121952-01	Easy-to-read presentation of the pallet management
155	Component Monitoring	06	1226839-01	Function for monitoring of components in regard to wear and overload

Overview
– Comparison of controls

Comparison of controls	TNC 620 NC SW 81760x-06	TNC 640 NC SW 34059x-09	iTNC 530 NC SW 60642x-04
Area of application	**Standard milling**	**High-end milling/turning**	**High-end milling**
• Simple machining centers (up to 5 axes + 1 spindle)	●	●	●
• Machine tools/machining centers (up to 18 axes + 2 spindles)	–	●	●
• Milling/turning operations (up to 18 axes + 2 spindles)	–	Option	–
Program entry			
• In HEIDENHAIN Klartext conversational programming language	●	●	●
• According to ISO	●	●	●
• CAD Import	Option	Option	Option
• CAD viewer	●	●	Option
• FK free contour programming	Option	●	●
• Extended milling and drilling cycles	Option	●	●
• Turning cycles	–	Option	–
NC program memory	2 GB	> 21 GB	> 21 GB
5-axis and high-speed machining	Option	Option	Option
Block processing time	1.5 ms	0.5 ms	0.5 ms
Input resolution and display step (standard/option)	0.1 µm/0.01 µm	0.1 µm/0.01 µm	0.1 µm/–
New design of the screen and keyboard	15-inch screen	15/19-inch screen	15/19-inch screen
Optimized user interface	●	●	–
Adaptive Feed Control (AFC)	–	Option	Option
Active Chatter Control (ACC)	Option	Option	Option
Dynamic Collision Monitoring (DCM)	–	Option	Option
KinematicsOpt	Option	Option	Option
KinematicsComp	–	Option	Option
Touch probe cycles	Option	●	●
Pallet Management	Option	●	●
Parallel axis function	●	●	–

● Standard
– Not available

HEIDENHAIN

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

www.heidenhain.de

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

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

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

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

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

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

HEIDENHAIN Technisches Büro Südost
83301 Traunreut, Deutschland
☏ 08669 31-1337

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 Victoria 3026, Australia
E-mail: sales@fcrmotion.com

BE **HEIDENHAIN N.V.**
1760 Roosdaal, Belgium
www.heidenhain.be

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

BR **HEIDENHAIN Brasil Ltda.**
04763-070 – São Paulo – SP, Brazil
www.heidenhain.com.br

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

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

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

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

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

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

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

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

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

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

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

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

HR Croatia → **SL**

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

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

IL **NEUMO VARGUS MARKETING LTD.**
Holon, 58859, Israel
E-mail: neumo@neumo-vargus.co.il

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

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

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

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

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

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

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

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

NZ **Llama ENGINEERING Ltd**
5012 Wellington, New Zealand
E-mail: info@llamaengineering.co.nz

PH **MACHINEBANKS' CORPORATION**
Quezon City, Philippines 1113
E-mail: info@machinebanks.com

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

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

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

RS Serbia → **BG**

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

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

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

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

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

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

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

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

UA **Gertner Service GmbH Büro Kiev**
02094 Kiev, Ukraine
www.heidenhain.ua

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

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

896140-29 · 7.5 · 04/2019 · CD · Printed in Germany