HEIDENHAIN

TNC 320

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

06/2018

This brochure describes the functions and specifications of the TNC 320 with NC software 771851-05.

Contents

The TNC 320...

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.

You'll also find this principle realized in the TNC 320: workshop-oriented programmability with graphic support, many field-proven cycles and an operational design that you are familiar with from other HEIDENHAIN controls.

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 320 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.

Simple operation
For simple work, such as face milling, you need not write a program on the TNC 320. It is just as easy to operate the machine manually by pressing the axis keys or—for maximum sensitivity—using an electronic handwheel.

Offline program creation
The TNC 320 can be programmed remotely just as well. Its Fast Ethernet interface guarantees very short transfer times, even of long programs.

The TNC 320 is compact and easy to read.
The TNC 320 is a compact but versatile contouring control for three servo axes and servo spindle. Two further servo axes are an option. Thanks to its flexible operation—workshop-oriented programmability with HEIDENHAIN Klartext programming or offline programming—and its scope of features, it is especially suited for use on universal milling, drilling, and boring machines for the following:

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

It also offers the applicable features both necessary and helpful for:

Universal milling machines

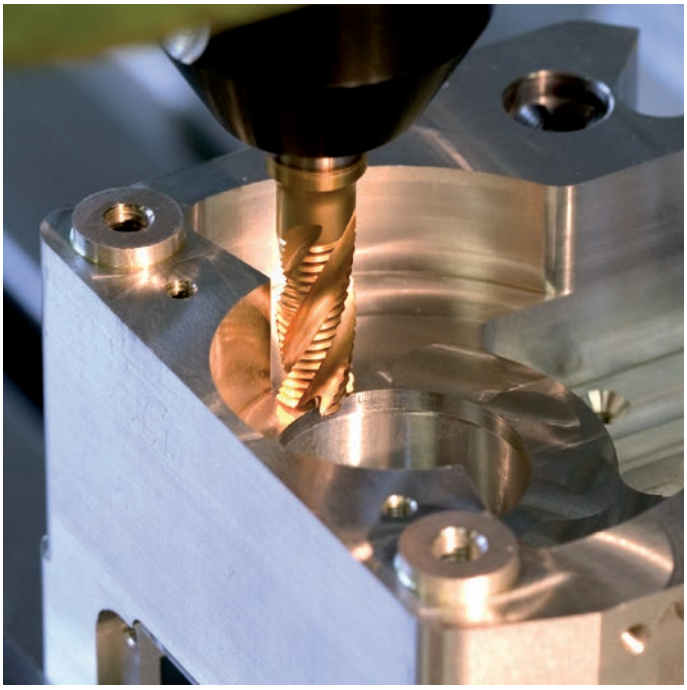
- Free contour programming
- Milling cycles for complex contours
- Fast presetting with HEIDENHAIN touch probes

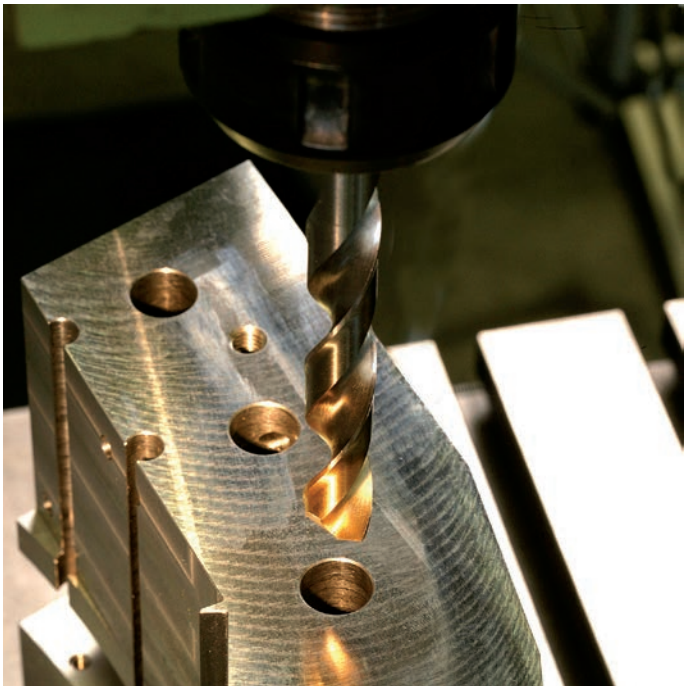
Drilling and boring machines

- Cycles for drilling, boring, and spindle alignment
- Cycles for Cartesian and polar point patterns
- Drilling oblique holes

Machines with parallel secondary axes

- Compensation of movement in the secondary axes U, V, W through the principal axes X, Y, Z
- Defining the principal and secondary axes in the NC program makes it possible to run programs on different machine configurations
- Movements of a parallel axis included in the position display of the associated principal axis (sum display)





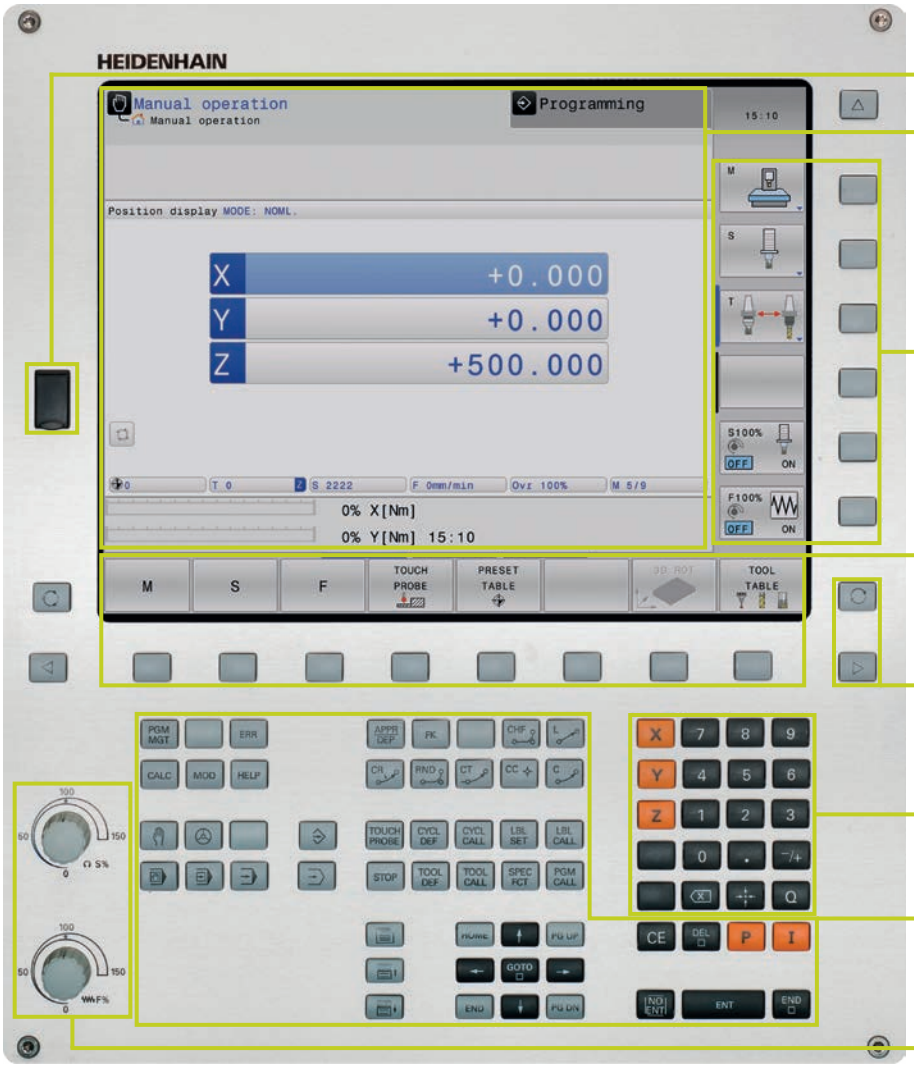
Well designed and user friendly
– The TNC 320 in dialog with the user

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

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

During the course of the program, status displays will always offer information to keep you up to date on tool position, the current program, active cycles and coordinate transformations, and other data. The TNC 320 even shows the current machining time.

The operating panel
As with all TNCs from HEIDENHAIN, the operating panel is oriented to the programming process. The well-thought-out arrangement of keys in a clear division into **function groups,** i.e. programming modes, machining modes, management/ TNC functions, and navigation, supports you during program input. Simple key assignments and easily understandable symbols or abbreviations clearly indicate each key's function. You use the **override potentiometers** to make delicate adjustments to the feed rate and spindle speed.



USB port for additional data storage or pointing devices

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

PLC function keys (soft keys) for machine functions

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

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

Axis-selection keys and **numeric keypad**

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

Override potentiometers for feed rate and spindle speed

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

Durable
The high-quality stainless steel design of the TNC 320 features a special protective 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.

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 and spindle speed.

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

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

Reliable
The elevated key bed of the machine operating panel (on versions with separate keyboard) prevents accidental actuation. LEDs serve for status display of each key by clearly indicating the active machine functions.

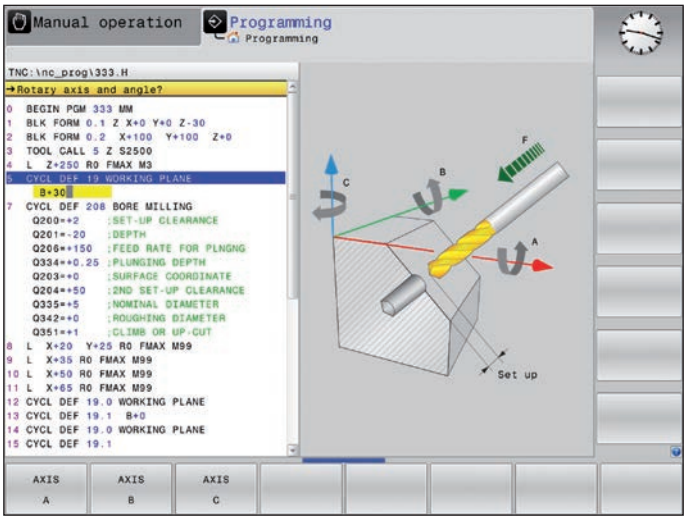
Well designed and user friendly
– The functional user interface

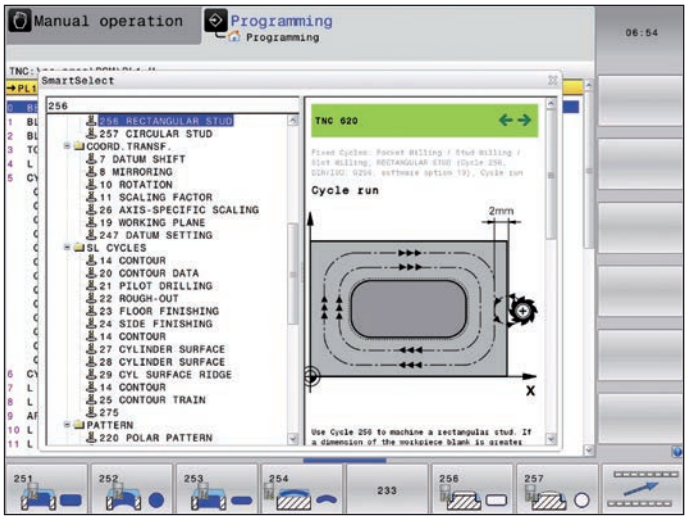
The combination of the straightforward and ergonomically designed keyboard and the well-designed screen layout are the essence of reliable and fatigue-free operation—principles that HEIDENHAIN has always represented. However, the TNC 320 also offers a number of features that make working with the control even easier and user-friendlier than ever.

Attractive view
The user interface of the TNC 320 has a modern appearance, with slightly 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 320 displays them in color-coded categories. A color-coded warning triangle is also displayed.

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 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.



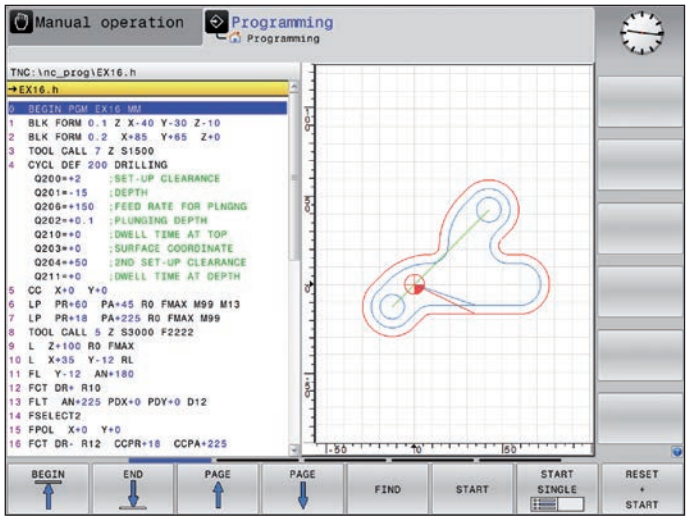


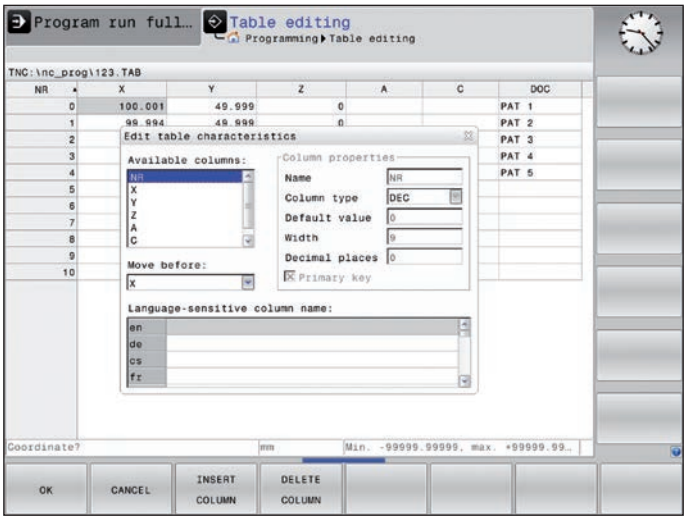
Color-structured programs
The content of a program line can be quite comprehensive: line number, program function, input values, comment. To help you always find your way even in complex programs, the individual program elements on the TNC 320 are shown in different colors. The color syntax highlighting 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 appearance, function and operation of the table editor are always the same.

Info line
In the info line, the TNC 320 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.





Minimize setup and non-machining time
– The TNC 320 makes setup easy

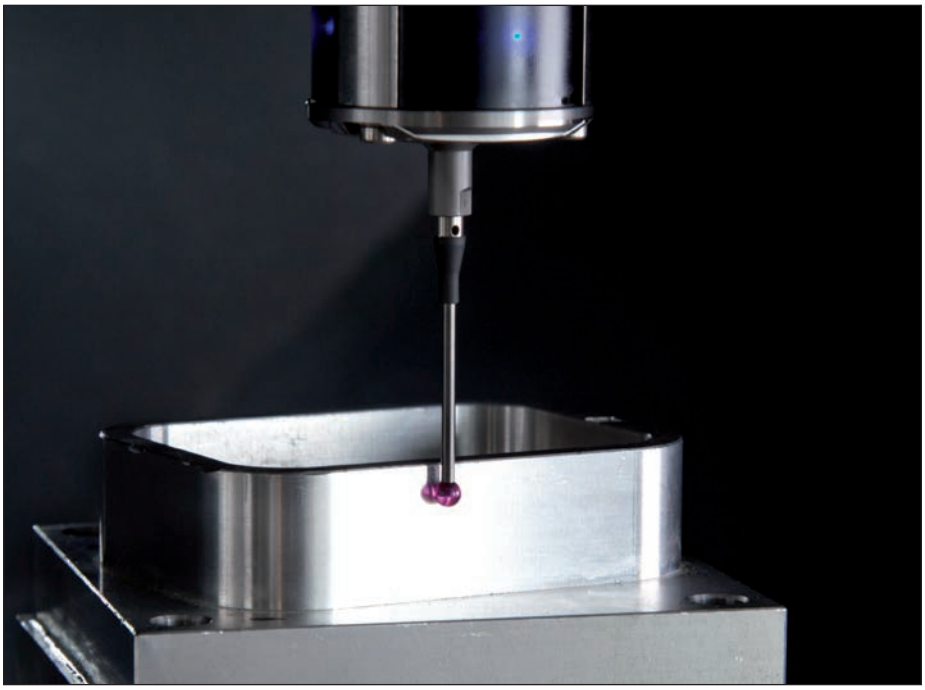
Before you can begin machining, you must first clamp the workpiece and set up the machine, find the position and orient the workpiece on the machine, and set the workpiece preset. Without support from the control this is often a time-consuming procedure, but it is indispensable. After all, any error 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.

Here the TNC 320 shows its strengths: With its practice-oriented setup features it supports the operator and helps to reduce non-machining time. Together with the **touch probes,** the TNC 320 offers various probing features for aligning, presetting, and measuring the workpieces.

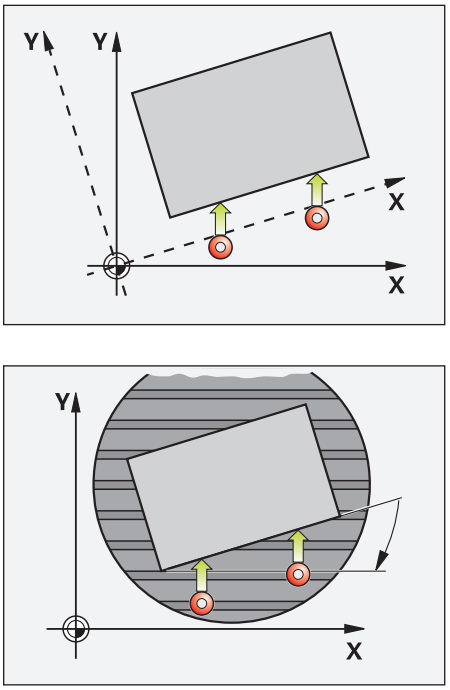
Delicate manual traverse
For setup, you can use the axis-direction keys to move the machine axes manually or in incremental jog. A simpler and more reliable way, however, is to use the electronic handwheels from HEIDENHAIN (see page 31). With the handwheels you are always close to the action, enjoy a 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 feed rate potentiometer to change the feed rate during probing. What is special about this option is that it does not influence accuracy.

Workpiece alignment
With HEIDENHAIN touch probes (see page 29) and the probing functions of the TNC 320, you can forgo any tedious manual alignment of the workpiece:
- Clamp the workpiece in any position.
- The touch probe determines the actual workpiece position by probing a surface, two holes, or two studs.
- The TNC 320 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.



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. Finding this point quickly and reliably reduces nonproductive time and increases machining accuracy.

The TNC 320 features probing cycles for automatic presetting. Once found, you can save these points
- in the preset manager,
- in a datum table, or
- by directly setting the displayed value.

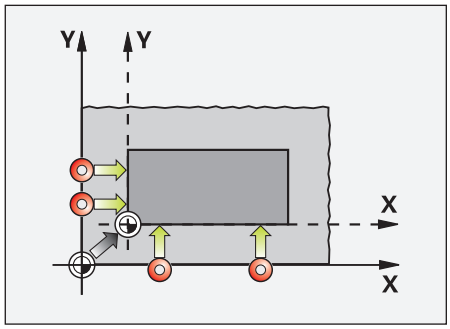
Preset management with the preset table
The preset manager makes flexible machining, shorter setup times, and increased productivity possible. In other words, it makes it much easier to set up the machine.

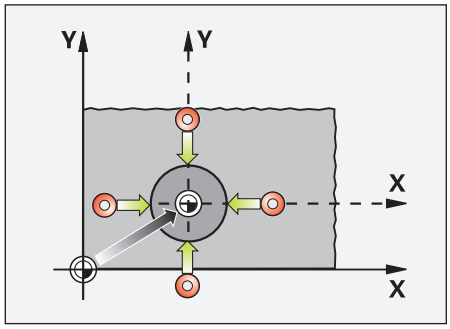
In the preset manager you can save **any number of presets** and assign an individual basic rotation to each one. To permanently save fixed presets in the machine working space, you can also write-protect individual lines.

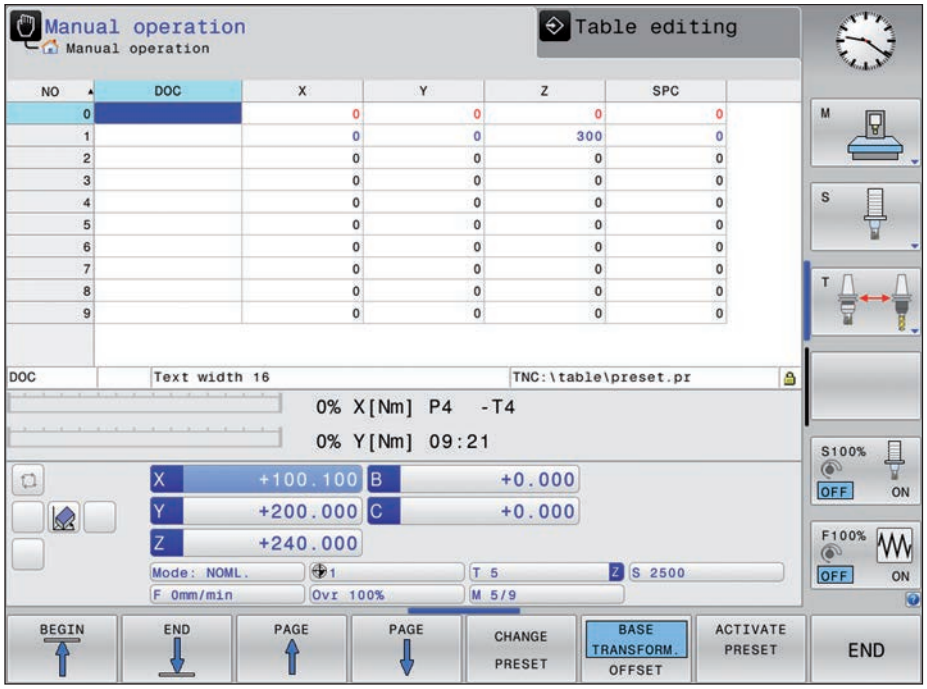
There are three ways to save presets rapidly in the preset manager:
- In the Manual mode by soft key
- By using the probing functions
- With the automatic probing cycles

Saving datums
In datum tables, you can save positions or values given 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





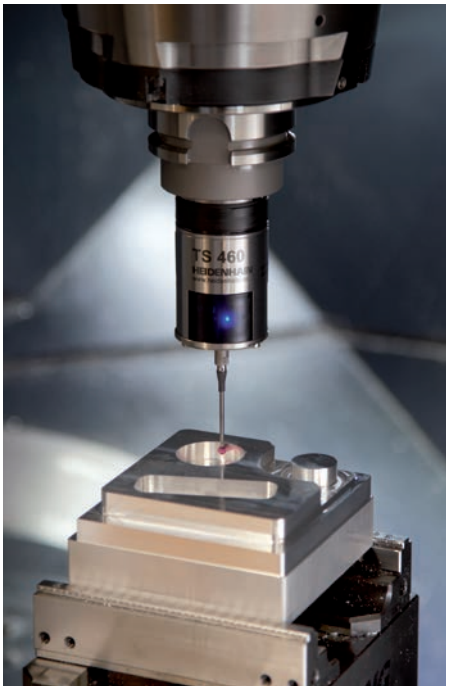


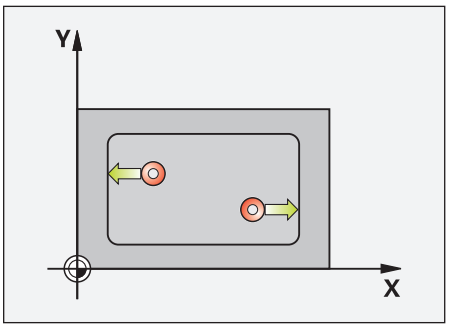
Minimize setup and non-machining time
– The TNC 320 manages and measures

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 320 is capable of controlling automated manufacturing processes. It has the functions required to manage and measure tools and inspect the machining process. It helps you reduce non-cutting time, increase productivity and improve production quality.

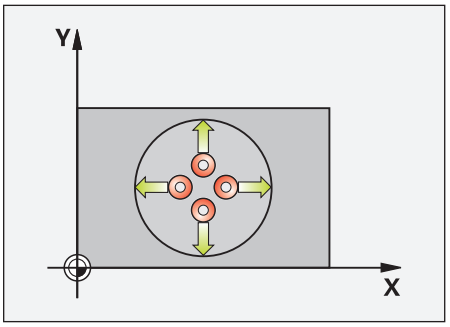
Inspecting workpieces for proper machining and dimensional accuracy
The TNC 320 features a number of measuring cycles for checking the geometry of the machined workpieces. For the measuring cycles you insert a touch probe from HEIDENHAIN (see page 29) into the spindle in place of a tool. This enables 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,
- ascertain the machining error trend.

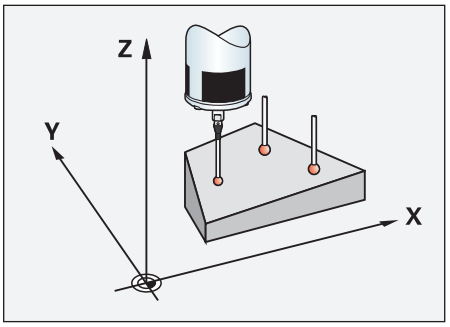




Length measurement

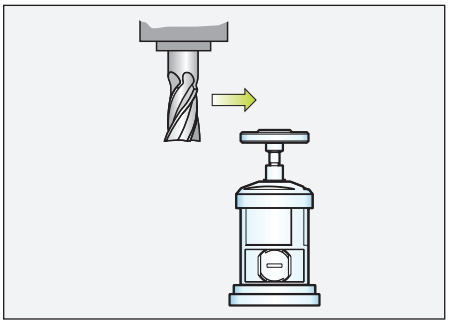


Measuring a circular pocket/hole

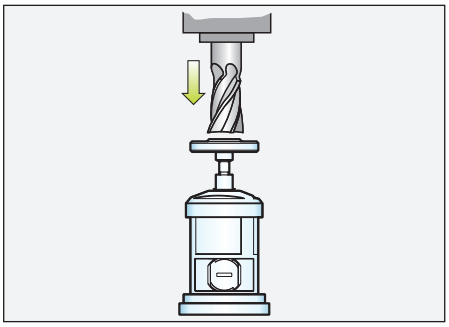


Measuring the angle of a plane

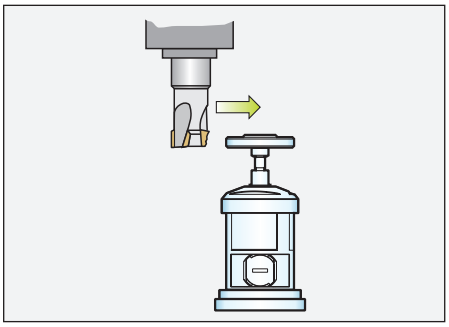
Tool measurement and automatic compensation of tool data
Together with the TT and TL touch probes (see page 30), the TNC 320 can automatically measure tools while they are in the machine. The TNC 320 saves the ascertained values of tool length and radius in the central tool file. By inspecting the tool 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 320 locks the tool and automatically inserts a replacement tool.



Measuring the tool radius



Measuring the tool length



Measuring tool wear

Tool management
For machining centers with automatic tool changers, the TNC 320 offers a central tool management function for any number of tools. The tool management is a freely configurable file and can therefore be optimally fitted to your needs. You can even have the TNC 320 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.*

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

Economical and productive
– Swivel head or rotary table controlled by the TNC (option)

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*
Programs for contours and holes on inclined surfaces are often very complex and require time-consuming computing and programming work. Here the TNC 320 helps you to save a great deal of programming time. You program the machining operation as usual in the working 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.

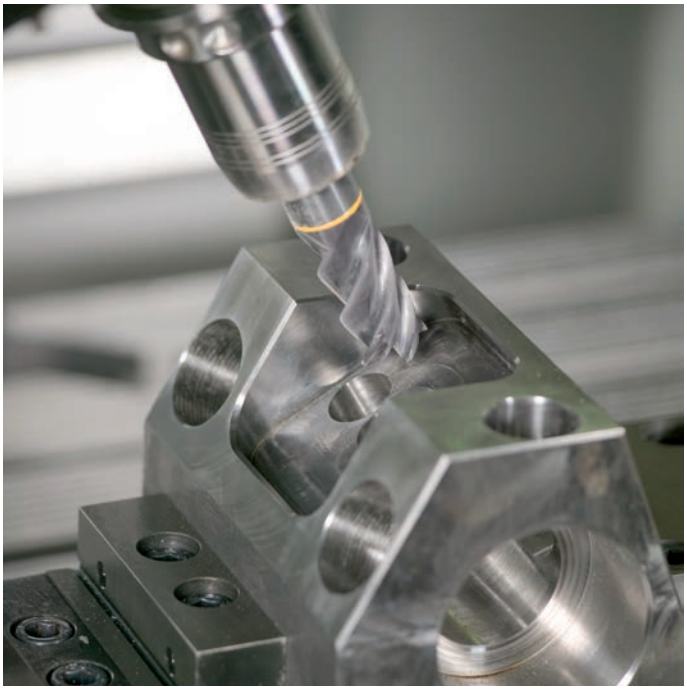
The PLANE feature makes it easy to define a tilted working plane: you can specify tilted working planes in seven different ways, depending on the information on the workpiece drawing. Clearly arranged support graphics assist you during input.

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*
With the TNC 320 it is quite easy to program contours (which consist of straight lines and arcs) on cylindrical surfaces using rotary and tilting tables: You simply program the contour as if the cylinder surface were unrolled. You enter a contour in two dimensions—as if in a plane—and the TNC 320 then calculates and machines the corresponding cylindrical contour.

The TNC 320 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

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



– Machining any contour slots with trochoidal milling

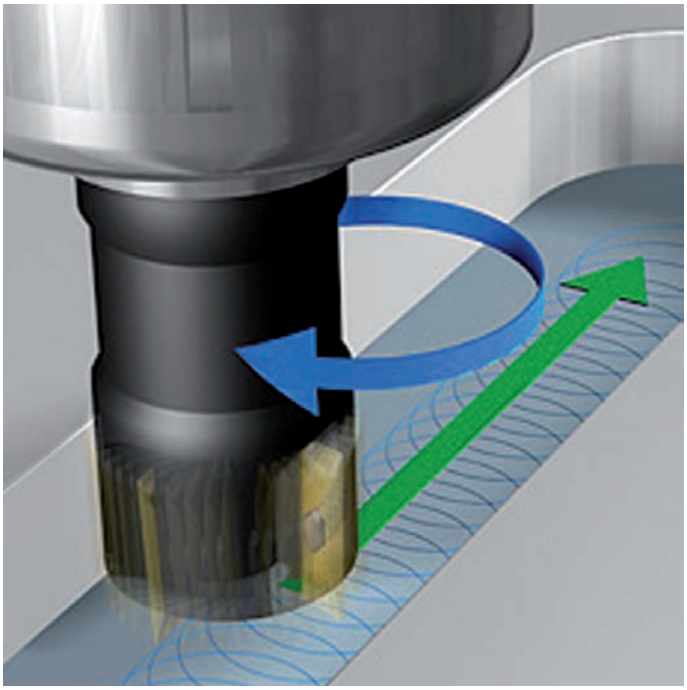
The benefit of trochoidal milling is its ultra-efficient machining of slots of all kinds. The roughing process is a circular motion superimposed on a forward linear motion. This procedure is referred to as trochoidal milling. It is used particularly for milling high-strength or hardened materials, where 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 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:

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



Programming, editing, testing
– The TNC 320 opens endless possibilities

The TNC 320 is just as universal in application as it is flexible in machining and programming.

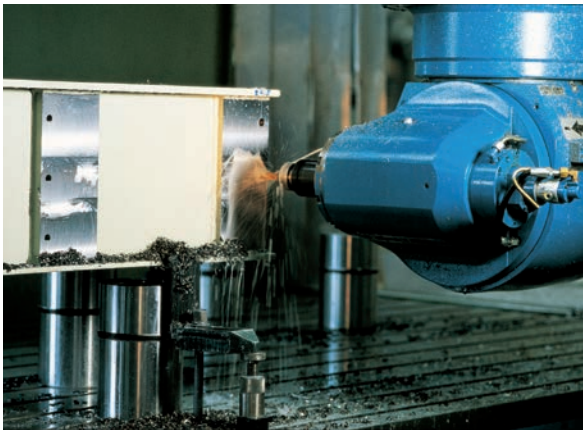
Positioning with Manual Data Input
You can start working with the TNC 320 even before writing a complete part program. Simply machine a part step by step—switching as you want between manual operation and automatic positioning.

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.

If you are used to **G-code programming,** however, the TNC 320 is still the right control for you. It displays soft-key rows dedicated to the most important G-code commands so that you can enter programs with G-codes directly. Or you connect a USB keyboard and use it to write the program.

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

Creating programs offline
The TNC 320 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 320 can also run programs that were written in ISO format.



– Graphic support in any situation

Programming graphics
The two-dimensional programming graphics give you additional security: while you are programming, the TNC 320 draws every entered traverse command on the screen. You can select among the plan view, side view, and front view.

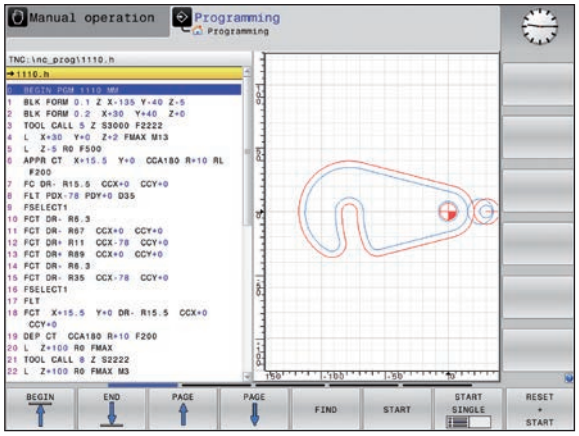
Test graphics
To play it safe before running a program, the TNC 320 can graphically simulate the machining of the workpiece. It can present different views of the simulation:

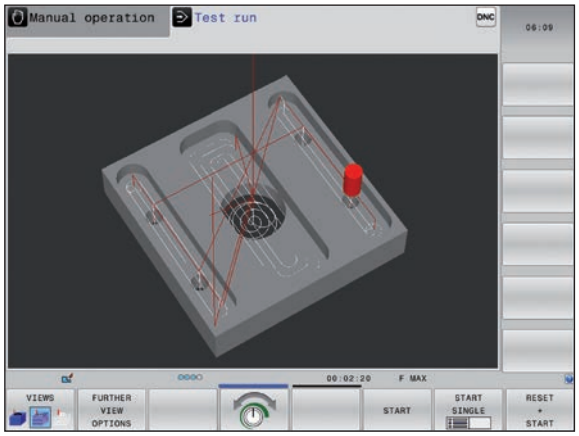
- As a plan view with different shades of depth
- In three projections
- In 3-D view

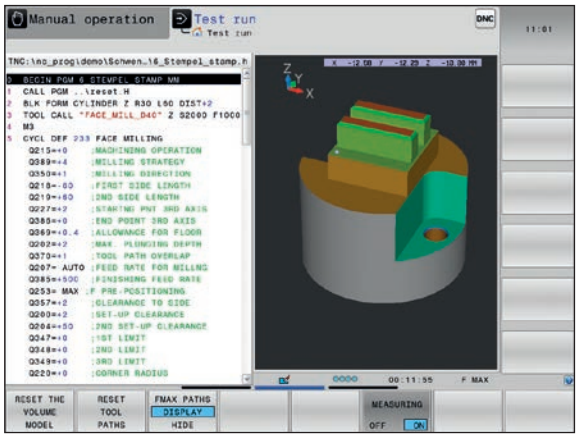
Details can be displayed with magnification. In addition, the TNC 320 indicates the calculated machining time in hours, minutes, and seconds.

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

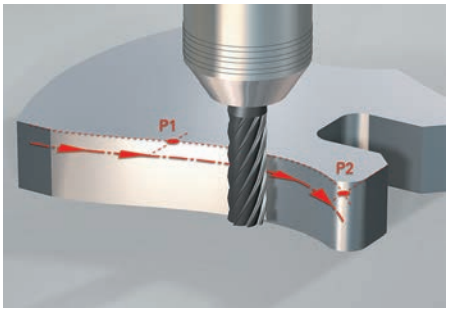
Two-dimensional contours are the bread and butter of a modern machine shop. Here the TNC 320 offers a variety of possibilities.

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

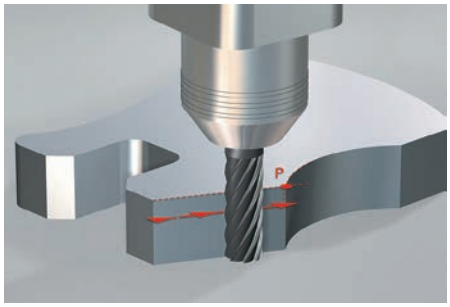
Straight and circular contour elements
To program a line segment, for example, simply press the key for linear traverse. The TNC 320 asks in Klartext format for all information required for a complete programming block, such as target coordinates, feed rate, cutter radius compensation, and machine functions. Appropriate path function keys for circular movement, chamfers, and corner rounding simplify your programming. To avoid surface blemishes during approach 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.

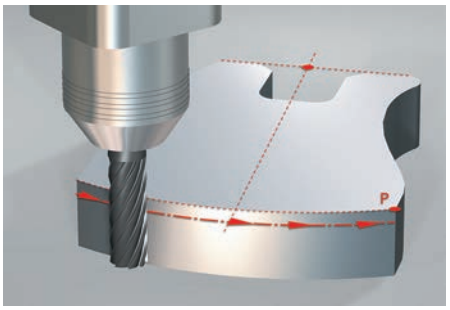
The TNC 320 can look ahead over a radius-compensated contour for up to 99 blocks to watch for back cutting and avoid contour damage such as can occur when roughing a contour with a large tool.



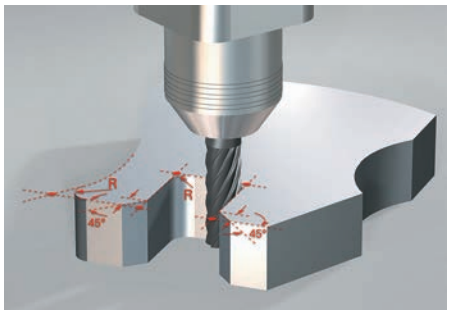
CT
Circular path with smooth, (tangential) connection with the preceding contour element, defined by end point



L
Straight line defined by its end point

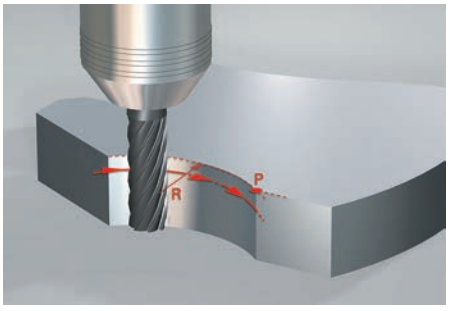


CC C
Circular path defined by its center point, end point, and rotational direction



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

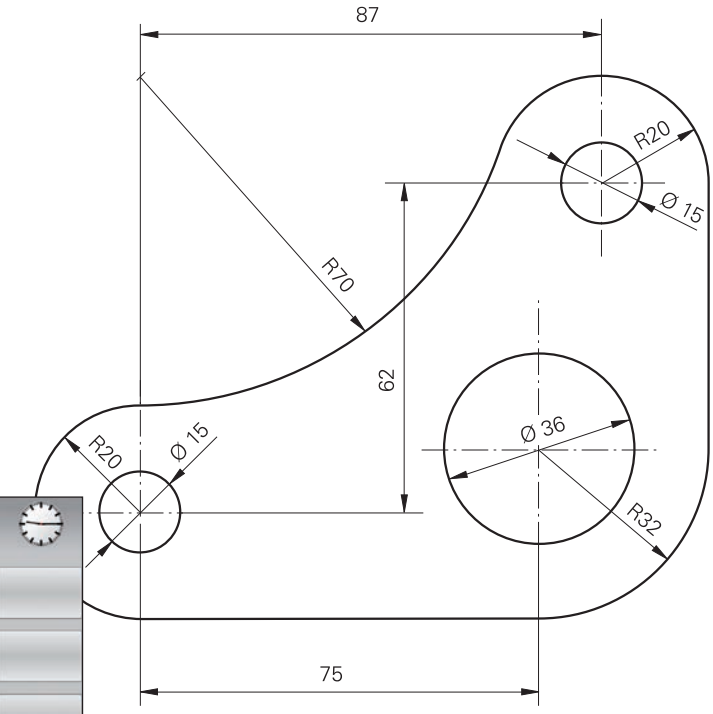
CHF
Chamfer: defined by the corner point and the chamfer length

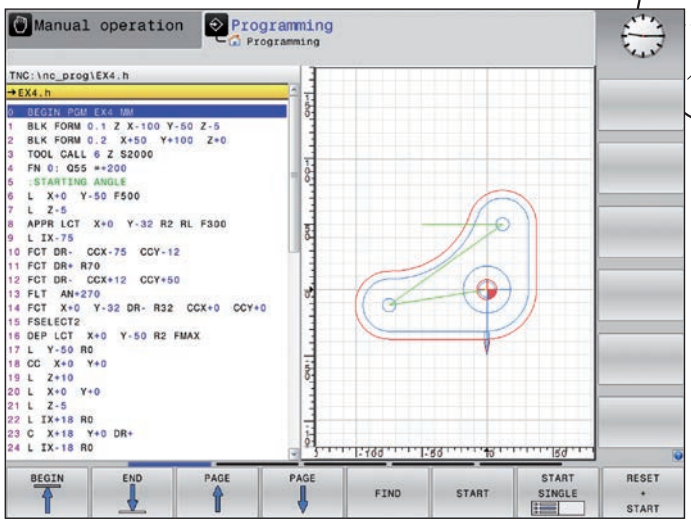


CR
Circular path defined by its radius, end point, and rotational direction

– Programming contours unconventionally

FK free contour programming
Not all workpieces are dimensioned for conventional NC programming. Thanks to FK, the 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 result in more than one mathematical solution, the helpful TNC 320 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

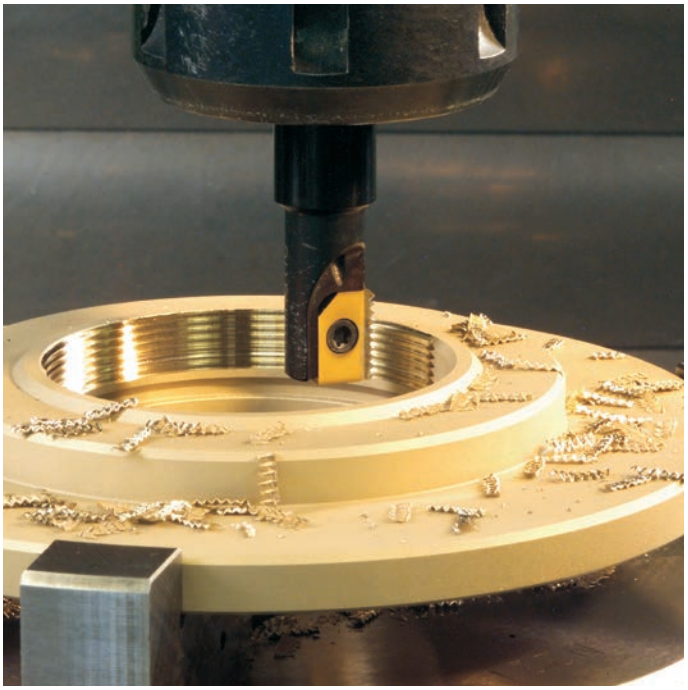
Frequently recurring operations that comprise several working steps are stored in the TNC 320 as cycles. You program them under conversational guidance and are supported by graphics that clearly illustrate the required input parameters.

Standard cycles
Besides the fixed cycles for drilling, tapping (with or without floating tap holder), thread milling, reaming, engraving, and boring, there are cycles 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
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.

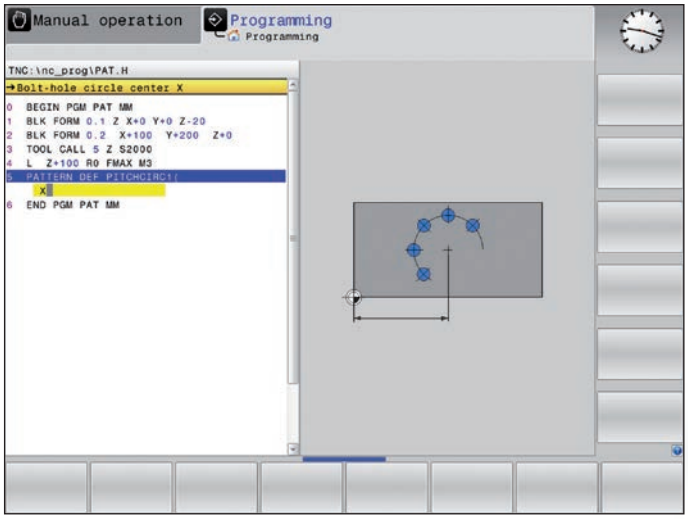
The TNC 320 maintains a **finishing allowance** on the wall and floor surfaces during roughing. When **roughing** with different tools, the control identifies material remaining in inside corners so that it can be cleared later with smaller tools. A separate cycle is used for milling to the finished dimension.



OEM cycles
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 320. 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 320 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 320, 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 execute 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 320 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.

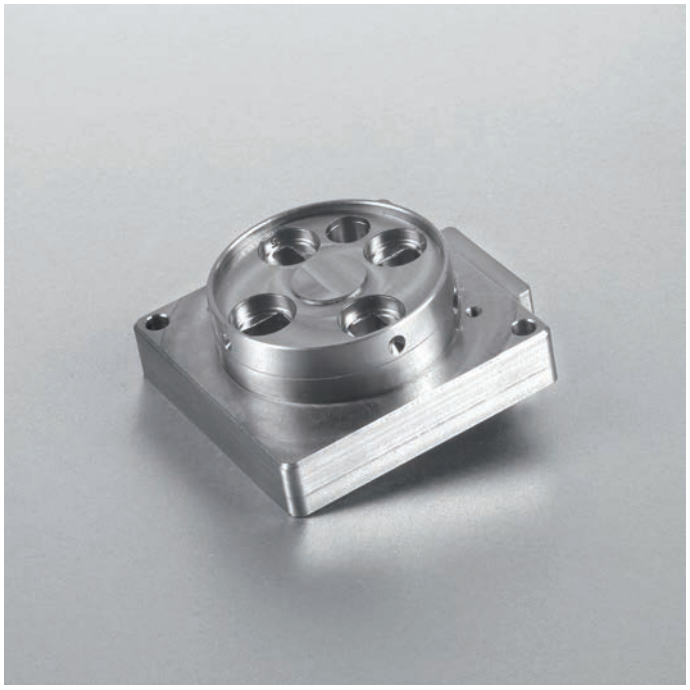
Program-section repeats and subprograms
Many machining operations repeat themselves either on the same workpiece or on different workpieces. Once you have programmed a detail, there is no reason to have to program it again. With its subprogramming feature, the TNC can save you a great deal of programming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

Of course you can also combine these programming techniques.



– 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 320 numerical control and TNC 320 programming station now feature TNCguide, a convenient help system that can show the user documentation in a separate window.

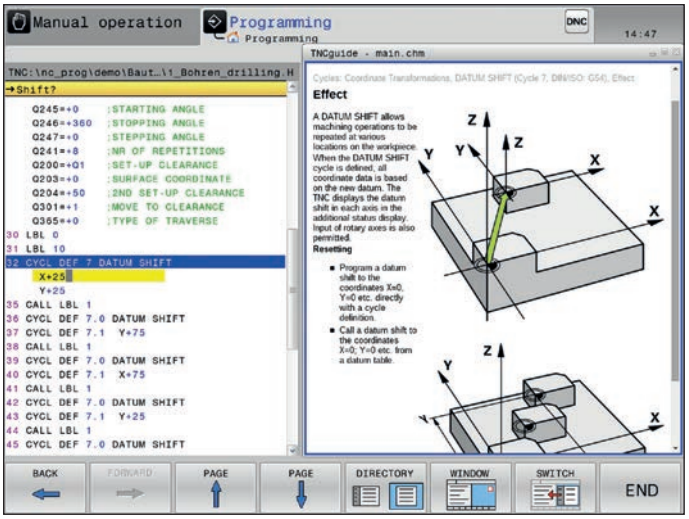
You can activate TNCguide by simply pressing the help key on the TNC keyboard or by clicking any soft key with a cursor in the shape of a question mark. You switch the cursor by simply clicking the help symbol that is always visible on the TNC screen.

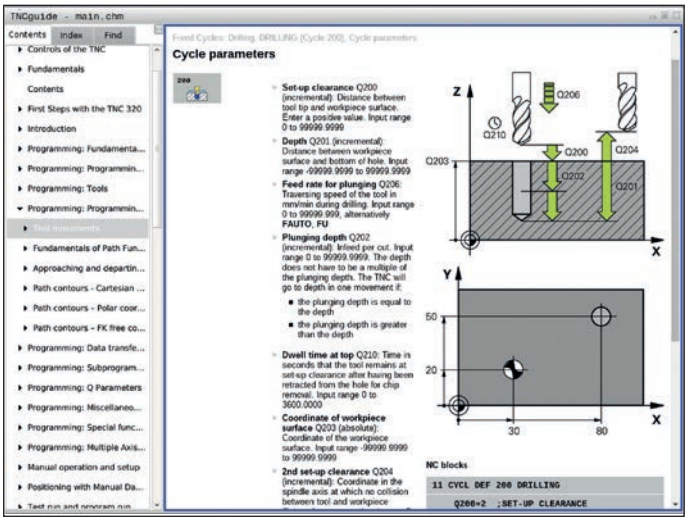
TNCguide usually displays the information in the immediate context of the element in question (context-sensitive help). This means that you immediately receive the relevant information. This function is particularly helpful with the soft keys. The method and effect of operation are explained in detail.

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 programming
- User's manual for cycle programming
- User's manual for ISO programming
- User's manual for the TNC 320 programming station (installed only with the programming station)





Open for communication
– The TNC 320 understands CAD files (option)

CAD viewer
The CAD viewer (standard feature) enables you to open 3-D CAD models and drawings directly on the TNC 320.

Various view options as well as rotation and zoom capabilities allow detailed visual control and analysis of your CAD data. You can also use the viewer to determine position values from a 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 in 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 320 detects your desired direction of machining and starts automatic contour detection. The TNC 320 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.

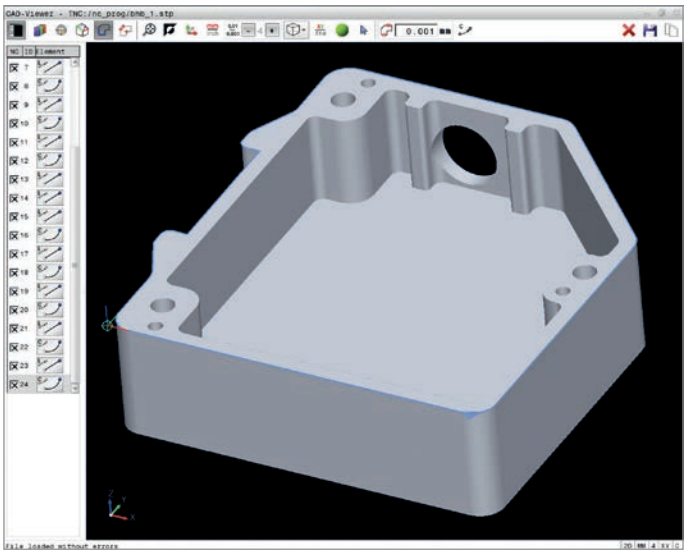
But you can also select **machining positions** and save them as point files, particularly in order to use drilling positions or starting points for pocket machining. This can be done very easily: using the mouse, simply select the desired area. In a pop-up window with a filter function, the TNC 320 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.

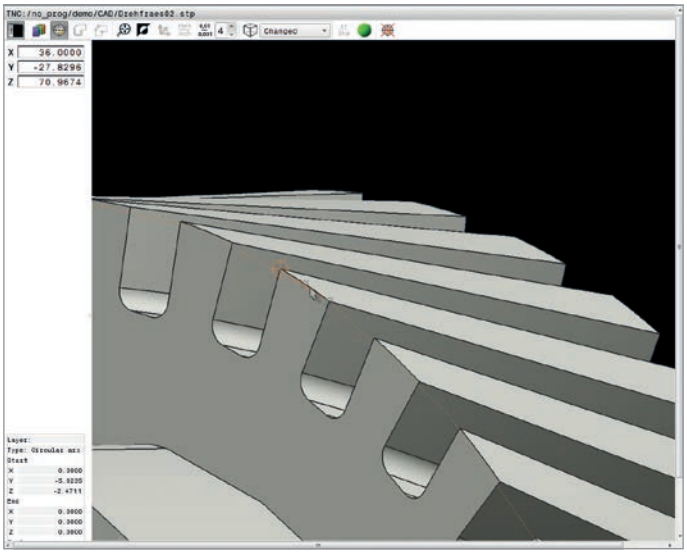
You can define the following locations as presets:

- 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



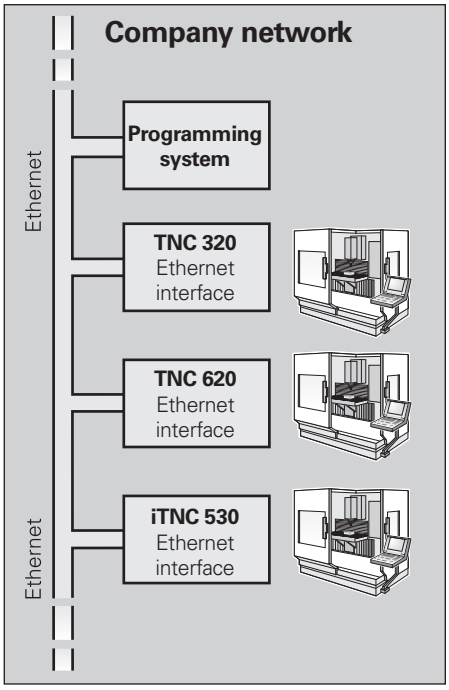
Display of a 3-D model in the CAD viewer

Open for communication
– Fast data transfer with the TNC

The networked TNC 320
The TNC 320 can be integrated into networks and connected with PCs, programming stations, and other data storage devices. Even in its standard version, the TNC 320 features a latest-generation Fast Ethernet interface in addition to its RS-232-C/V.24 data interface. The TNC 320 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 Mbps guarantees very short transfer times.

The transmitted programs are saved in the internal memory of the TNC 320 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, you can
- transfer remotely stored part programs and tool or pallet tables in both directions and
- make backups.

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

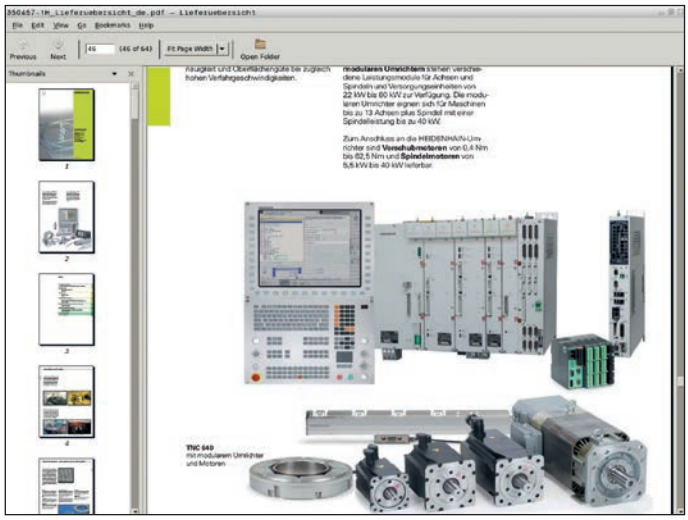
The integrated PDF viewer enables you to open PDF files directly on the control. The PDF format is a widely used data format that can be generated out of a great variety of applications. This enables you to easily view work instructions, drawings, or other information in the TNC.

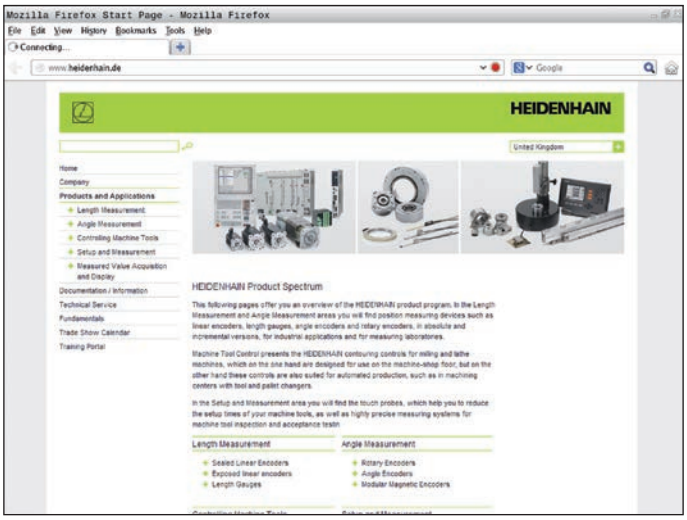
The integrated browser now lets you connect the TNC to the Internet and access it directly from the control.

The following further file formats can also be opened directly on the TNC with the appropriate programs, and some of them can be edited:
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

An operating panel with an integrated touchpad or an external USB pointing device is required for operation.





Open for external information
– The TNC 320 programming station

Why a programming station?
It's well known that it is easy to create part programs on a TNC 320 at the machine, even while another part is being machined. Nevertheless, short reloading times and other machining tasks can often hinder any prolonged or concentrated programming work. With the TNC 320 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 Klartext or G-code programs for the TNC 320 with the programming station substantially reduces machine idle times. 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
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 TNC 320 programming station
Because the TNC 320 programming station is based on the same software as the TNC 320, it is ideally suited for apprentice and advanced training. Programming and program testing function in exactly the same way as they do on the machine. This gives the trainee the experience needed to enable him to safely operate the machine later.

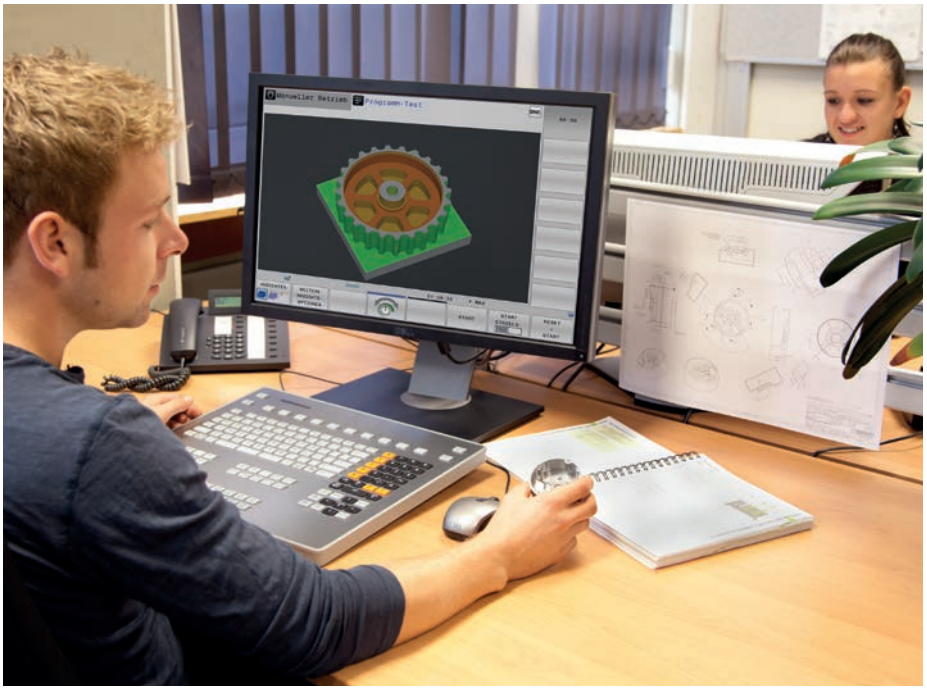
Since the TNC 320 can be programmed in Klartext and with G codes, the TNC 320 programming station can also be used in schools for TNC programming training.

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** contains all functions of the TNC 320, and permits short programs to be saved. It is programmed via the PC keyboard.

On the version with the TE 520B **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 320's most important dialog initiation keys.



Programming station with TNC operating panel

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

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 320, 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.

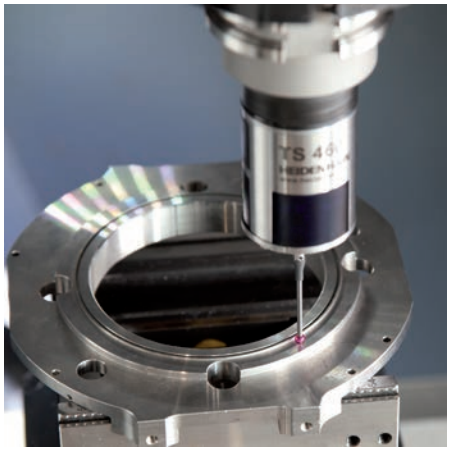
* Machine and TNC must be specially adapted by the machine tool builder.

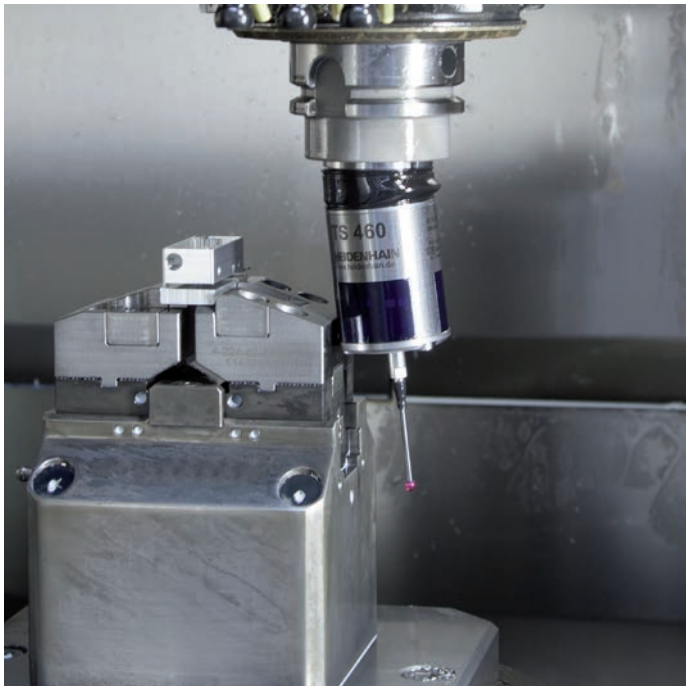
Touch probes with **cable connection for signal transmission** for machines with manual tool change, as well as for grinding machines and lathes:

TS 260 – New generation, axial or radial cable

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

TS 460 – New-generation standard touch probe for radio and infrared transmission with compact dimensions, energy-saving mode, optional collision protection, and thermal decoupling
TS 444 – Battery-free power supply through integrated air turbine generator from compressed air, for infrared transmission, with compact dimensions
TS 740 – High probing accuracy and repeatability, low probing force, with infrared transmission





TS 460 with collision protection

TS 460

SE 660

More information about workpiece touch probes is 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 periodic inspection of the tool for wear and breakage, as well as the shape of each tooth, are necessary. HEIDENHAIN offers the triggering TT 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 rotating or stationary tool, e.g. during individual tooth measurement, the contact plate is deflected and a trigger signal is transmitted directly to the TNC 320.

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.

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, in addition to measuring the tool length and radius, can detect errors in the shape of individual teeth.

TL Micro

TT 460

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

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 HR 510, HR 520, and HR 550 portable handwheels are particularly helpful when you work in close proximity 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.

Expanded feature content of 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 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 defined by the machine tool builder

HR 550

Overview
– User functions

User functions	Standard	Option	
Short description	•		Basic version: 3 axes plus closed-loop spindle
		○	1st additional axis for 4 axes plus open-loop or closed-loop spindle
		○	2nd additional axis for 5 axes plus open-loop or closed-loop spindle
Program entry	•		HEIDENHAIN Klartext
	•		ISO via soft keys or via USB keyboard
Position information	•		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)
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
Machining with a rotary table		8	Programming of cylindrical contours as if in two axes
		8	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	•		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
Fixed cycles	•		Cycles for drilling, pecking, reaming, boring, counterboring, conventional and rigid tapping
	•		Cycles for milling internal and external threads
	•		Rectangular and circular pockets
	•		Face milling
	•		Cycles for clearing level and inclined surfaces
	•		Multioperation machining of straight and circular slots
	•		Multi-operation machining of rectangular and circular pockets
	•		Cartesian and polar point patterns
	•		Contour train, contour-parallel contour pocket
	•		OEM cycles (special cycles developed by the machine tool builder) can be integrated
	•		Engraving cycle: Engrave text or numbers in a straight line or on an arc

User functions	Standard	Option	
Coordinate conversions	•	8	Shifting, rotating, mirroring, scaling (axis-specific) Tilting the working plane, PLANE function
Q parameters Programming with variables	• • • • •		Mathematical functions =, +, –, *, /, sin α, cos α, tan α, arc sin, arc cos, arc tan, a^n, e^n, ln, log, $\sqrt{a}$, $\sqrt{a^2 + b^2}$ Logical operations (=, = /, <, >) 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
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 320 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
Test graphics Display modes	• • •		Graphic simulation before a program run, even while another program is running Plan view / projection in 3 planes / 3-D view, also in tilted working plane / 3-D line graphics 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	• •		Graphic simulation during real-time machining 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
Touch probe cycles	• • • •		Calibrating the touch probe Compensating workpiece misalignment 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 CAD models on the TNC

Overview
– Options
– Accessories

Option number	Option	As of NC software 771851-	ID	Comment
0 **1**	Additional axis	01	354540-01 353904-01	• 1st additional axis for 4 axes plus open-loop or closed-loop spindle • 2nd additional axis for 5 axes and open-loop or closed-loop spindle
8	Advanced function set 1	01	536164-01	**Machining with a rotary table** • Programming of cylindrical contours as if in two axes • Feed rate in distance per minute
		01		**Interpolation** • Circular in 3 axes with tilted working plane **Coordinate transformation** • Tilting the working plane • PLANE function
18	HEIDENHAIN DNC	01	526451-01	Communication with external PC applications over COM component
42	CAD import	01	526450-01	Importing contours from 3-D models
93	Extended tool management	02	676938-01	Extended tool management

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 260:** touch trigger probe with cable connection, or • **TS 460:** touch trigger probe with radio or infrared transmission, or • **TS 444:** touch trigger probe with infrared transmission, or • **TS 640:** touch trigger probe with infrared transmission, or • **TS 740:** touch trigger probe with infrared transmission
Tool measurement	• **TT 160:** 3-D touch trigger probe, or • **TT 460:** 3-D touch trigger probe with radio or infrared transmission • **TL Nano:** laser system for contact-free workpiece measurement, or • **TL Micro:** laser system for contact-free workpiece measurement
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
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

– Specifications

Specifications	
Components	• Main computer with TNC keyboard and integrated 15.1-inch TFT color flat-panel display with soft keys
Operating system	• HEROS 5 real-time operating system for machine control
Memory	• 1.8 GB (on CFR compact flash memory card)
Input resolution and display step	• Linear axes: to 0.1 µm • Angular axes: to 0.0001°
Input range	• Maximum 99999.999 mm or 99999.999°
Interpolation	• Linear in 4 axes • Circular in 2 axes • Circular in 3 axes with tilted working plane • Helical: superimposition of circular and straight paths
Block processing time	• 3 ms (3-D straight line without radius compensation)
Axis feedback control	• Position resolution: Signal period of the position encoder/1024 • Cycle time of position controller: 3 ms
Range of traverse	• Maximum 100 m
Spindle speed	• Maximum 100 000 rpm (analog speed command signal)
Error compensation	• Linear and nonlinear axis error, backlash, reversal peaks during circular movements, thermal expansion • Static friction, reversal error, sliding friction
Data interfaces	• RS-232-C/V.24 max. 115 kbps • Extended data interface with LSV2 protocol for remote operation of the TNC 320 over the data interface with the HEIDENHAIN software TNCremo or TNCremoPlus • Gigabit Ethernet interface 1000BASE-T • 3 x USB (1 x front USB 2.0; 2 x back panel USB 3.0)
Diagnostics	• Fast and simple troubleshooting through integrated diagnostic aids
Ambient temperature	• Operation: 0 °C to +45 °C • Storage: –20 °C to +60 °C

HEIDENHAIN

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

www.heidenhain.de

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

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

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

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

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

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

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

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

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

AU **FCR MOTION TECHNOLOGY PTY LTD**
Laverton North Victoria 3026, Australia
E-mail: sales@fcrmotion.com

BE **HEIDENHAIN NV/SA**
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

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

1113511-23 · 7.5 · 06/2018 · H · Printed in Germany