HEIDENHAIN

TNC 320

Compact Contouring Control for Milling, Drilling and Boring Machines

www.heidenhain.com/cnc-controls

Contents

The TNC 320...

The functions and specifications described in this brochure apply to the TNC 320 with NC SW 771851-18.

Compact and versatile
The right control for milling, drilling and boring machines

For more than 45 years, TNC controls from HEIDENHAIN have proven themselves every day on milling machines, machining centers, and drilling machines. Over this period, they have been continually further developed, with focus placed on an operating approach tailored to the needs of the user.

These principles can also be found in the TNC 320: shop-oriented programmability with graphical support, many practical cycles, and an operational design that you are familiar with from other HEIDENHAIN controls.

Shop-oriented programming
You program conventional milling and drilling operations right at the machine, in Klartext—the dialog-guided, shop-oriented programming language from HEIDENHAIN. With helpful dialog guidance and informative visual aids, the TNC 320 provides optimal programming support.

For standard machining operations and even complex applications, a wealth of practical machining and coordinate transformation cycles are at your beck and call.

Easy to operate
When it comes to simple machining tasks, such as face milling, you don't need to write a program on the TNC 320. That's because it makes manual control equally easy—either by means of the axis keys or, for maximum sensitivity, with an electronic handwheel.

Offline programming
The TNC 320 is easy to program remotely as well. Its Ethernet interface ensures extremely short upload times, even for 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—shop-oriented programmability with HEIDENHAIN Klartext format or offline programming—and its scope of features, it is especially 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

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

Universal milling machines
• Free contour programming
• Milling cycles for complex contours
• Rapid preset setting with HEIDENHAIN touch probes

Drilling and boring machines
• Drilling and boring cycles
• Cycles for circular and linear point patterns
• Drilling of 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 enables execution on different machine configurations
• Movements of parallel axes included in the position display of the associated principal axis (sum display)
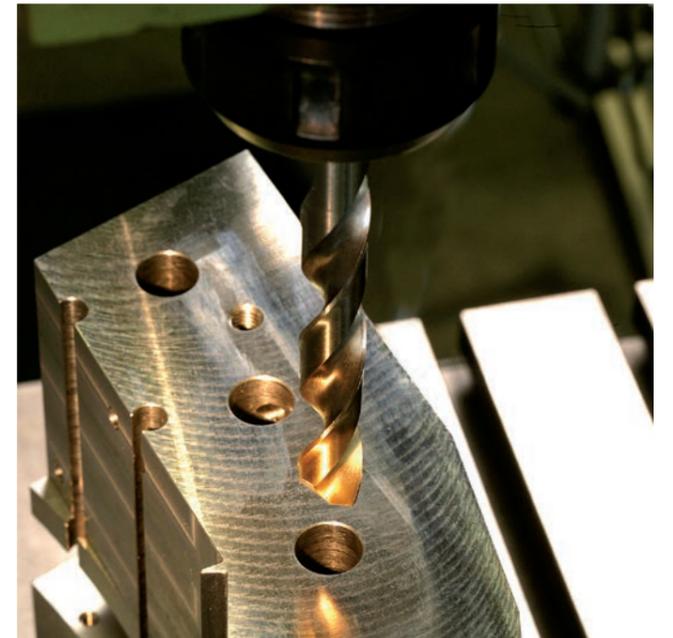
Intuitive and user-friendly
The TNC 320 in dialog with the user

The screen
In a clear format, the 15-inch screen displays all of the information you need for programming, operating, and checking the control and the machine, including program blocks, tips, and error messages. More information is provided through graphic support during program entry, test run, and actual machining.

You can also use a split screen to display your NC blocks on one side and the graphical image or the status display on the other.

While the program is running, status displays are available that inform you about the tool position, current program, coordinate transformations, and more. The TNC 320 even displays the current machining time.

The operating panel
Like with all of the TNCs from HEIDENHAIN, the operating panel was designed with the programming process in mind. Program entry is aided by the thoughtful arrangement of keys into function groups for programming operating modes, machine operating modes, management/TNC functions, and navigation. Intuitive key assignment, readily understood symbols, and abbreviations unmistakably identify the various functions. With the override potentiometers, you can make fine adjustments to the feed rate and spindle speed.
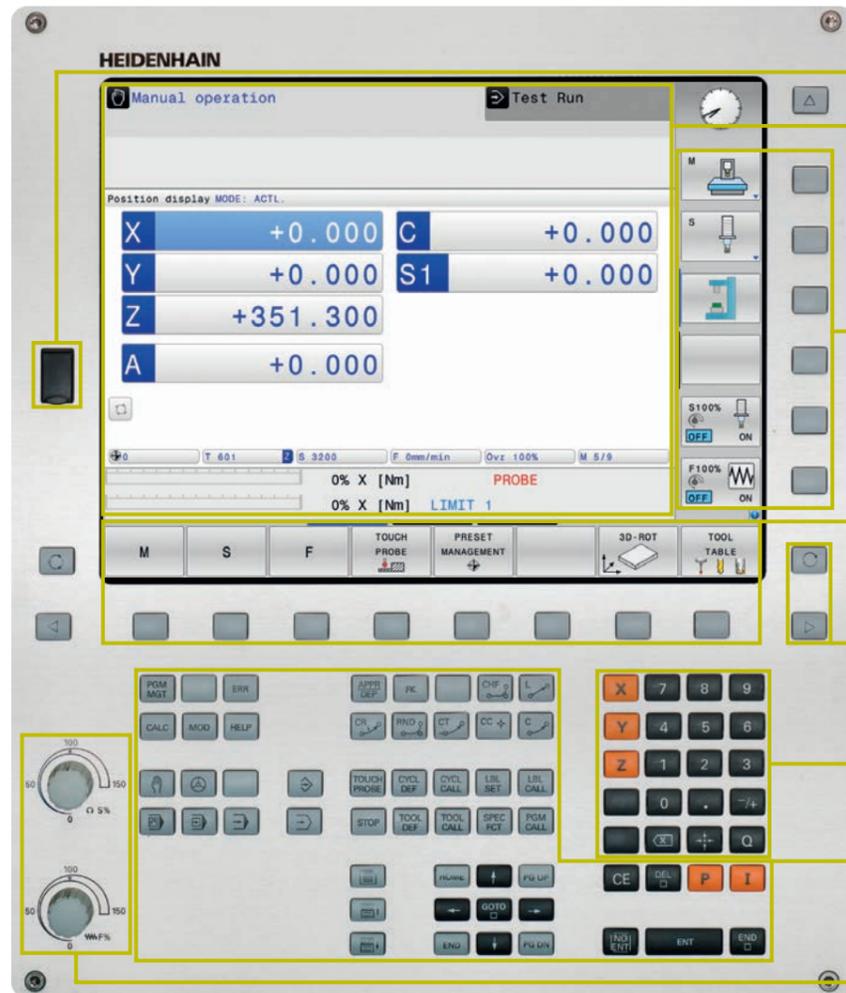
USB port for additional data storage or other pointing devices

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 operating 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 modern design. Judge for yourself:

Durable
The high-quality stainless-steel design of the TNC 320 features a special protective coating, making it particularly resistant to contamination and wear.

Smooth
The rectangular, slightly rounded keys are pleasant to the touch and reliable in operation. Their abrasion-resistant labeling can withstand extreme shop conditions.

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

Sensitive
With the handy control knobs, you can make fine adjustments to 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.

Highly practical user interface

The key to safe and low-fatigue work is having an intuitive screen layout and an ergonomic, well-arranged keyboard. Although HEIDENHAIN has been adhering to these principles for a long time, the TNC 320 offers a number of features that make working with the control even simpler and easier than before.

Leading-edge design
The user interface of the TNC 320 boasts a sleek outer appearance with gently rounded edges, color gradients, and a pleasingly uniform font. The different screen areas are clearly delineated, and the operating modes are indicated by special operating mode symbols.

To help you evaluate the priority of error messages, the TNC 320 displays them in color-coded categories. A color-coded warning triangle is also displayed.

The smartSelect functions overview
With dialog support, you can now readily select functions that were previously available only via soft key structures. As soon as you open smartSelect, you are shown a tree structure with all of the subordinate functions that can be defined in the control's current operating state. The TNC also displays integrated help on the right-hand side of the smartSelect window. When you make a selection with the cursor or per mouse click, you immediately receive detailed information about the function. The smartSelect feature is also available for the definition of machining cycles, touch probe cycles, and special functions (SPEC FCT), and for parameter programming.

Color-structured programs
The content of a program line can be quite extensive:
• Line number
• Program function
• Input value
• 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. This allows you, for example, to see at a glance input values are editable.

Universal table editor
Regardless of whether you are editing a tool, preset, or pallet table, the layout, method of operation, and functions of the table editor are always the same.

Info line
In the info line, the TNC 320 helps you orient yourself by displaying the respective status of the submode.

MOD function
The additional MOD operating mode offers a wide variety of possible settings in a standardized layout, regardless of the operating mode.

User administration
The user administration feature of the TNC 320 lets you define various roles and access rights for users, ensuring that each user can act only within his assigned rights. This helps prevent the unintentional or unauthorized deletion of files and system file content. Many functions are also available only with the corresponding permissions. Consequently, the user administration feature of the TNC 320 not only increases data security but also promotes operating safety.

8
9
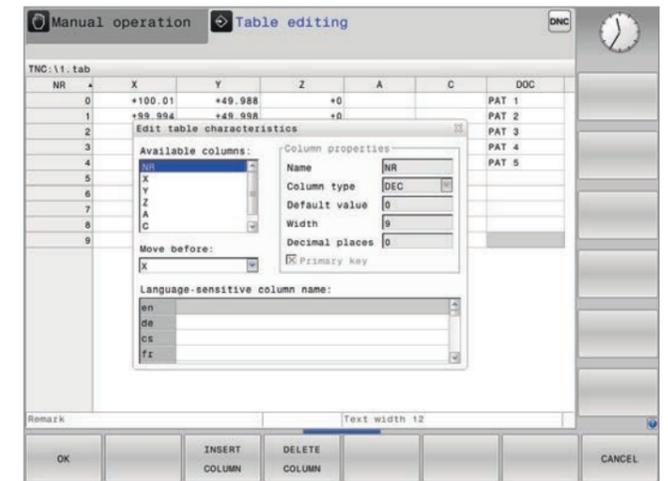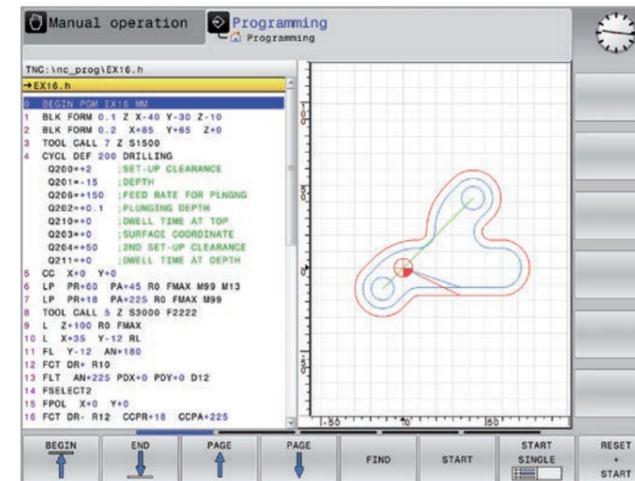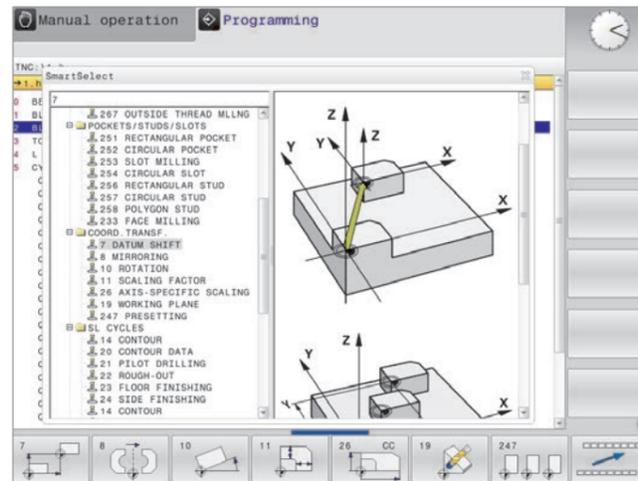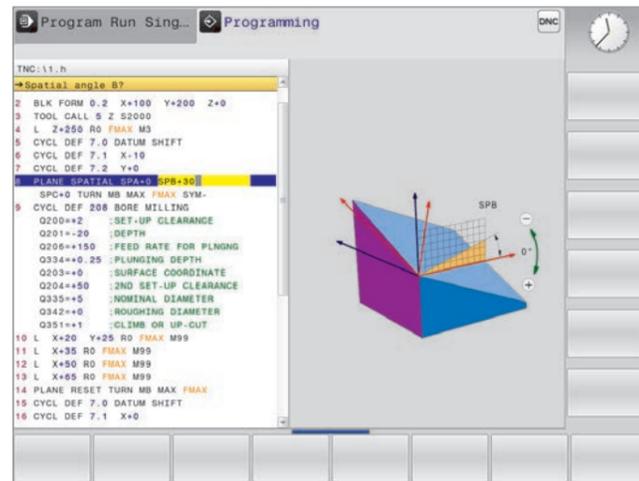
Minimize setup and non-machining time
The TNC 320 makes setup easy

Before machining can begin, the workpiece must first be clamped, the machine must be set up, the position of the workpiece in the machine must be determined, and the preset must be set. Without support from the control, this is often a time-consuming but indispensable procedure because any error directly reduces the machining accuracy. Particularly in small and medium-sized production runs, as well as in the case of very large workpieces, setup times become a significant factor.

Here the TNC 320 shows its strengths: with its practice-oriented setup features it assists the user and helps to reduce non-machining time. In conjunction with the touch probes, the TNC 320 offers numerous probing cycles for automatic workpiece setup, setting the preset, and measuring workpieces and tools.

The TNC 320 supports L-shaped styli in the probing cycles, which are used to quickly and easily probe undercuts.

Fine manual axis control
During setup, you can use the axis-direction keys to move the machine axes manually or with an incremental jog. A simpler and more reliable way, however, is to use electronic handwheels from HEIDENHAIN (see page 32). With these 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 speed
In many cases, the workpiece must be probed at unseen positions or in cramped spaces, and the standard probing feed rate is usually too fast for this. In such situations you can use the override potentiometer to change the feed rate during probing without affecting the accuracy.

Setting presets
A preset allows you to assign a defined value displayed by the TNC to any position on the workpiece. Finding this point quickly and reliably reduces non-productive time and increases machining accuracy.

The TNC 320 features probing cycles for automatic presetting. Ascertained presets can be saved as follows:
• Using the preset management
• In a datum table
• Through direct setting of the display

Workpiece alignment
Touch probes from HEIDENHAIN (see page 30) and the probing functions of the TNC 320 keep you from having to perform time-consuming workpiece alignment:
• You start by simply clamping the workpiece in any position.

• The touch probe then ascertains the actual workpiece position by probing a surface, two holes, or two studs.
• The TNC 320 subsequently compensates for the misalignment by performing a "basic rotation," in which either the NC program is executed at the ascertained angle of rotation or the rotary table is turned in order to correct the misalignment.

The TNC 320 offers you a comprehensive package of cycles for setting up the machine tool:
• Manual, automatic, and semiautomatic cycles for presetting and for compensating for misalignments in two or three dimensions
• Automatic probing cycle for repeating a measurement along a line

Preset management with the preset table
Preset management enables flexible machining, shorter setup times, and higher throughput. Setting up your machine is greatly simplified.

In preset management, 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's workspace, you can also write-protect individual lines.

There are three possibilities for rapidly saving the presets:
• Via a soft key in Manual mode
• Via the probing functions
• With the automatic probing cycles
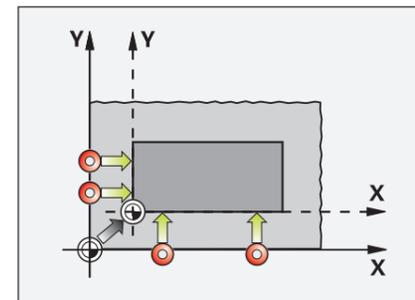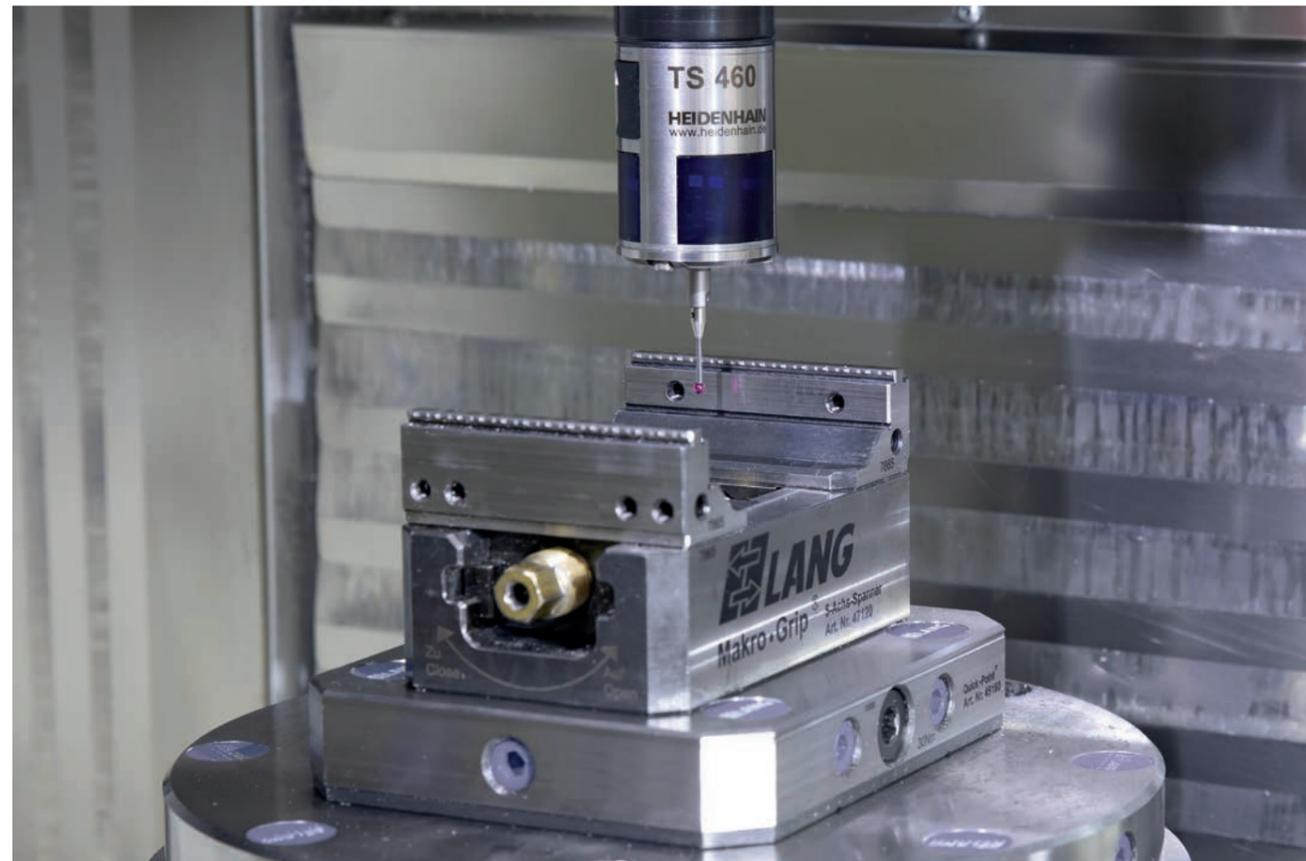
Setting a preset
on a corner, for example, or in the center of a circular stud
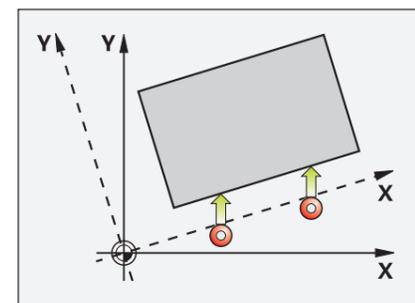
Workpiece misalignment compensated for
by rotating the coordinate system or turning the table
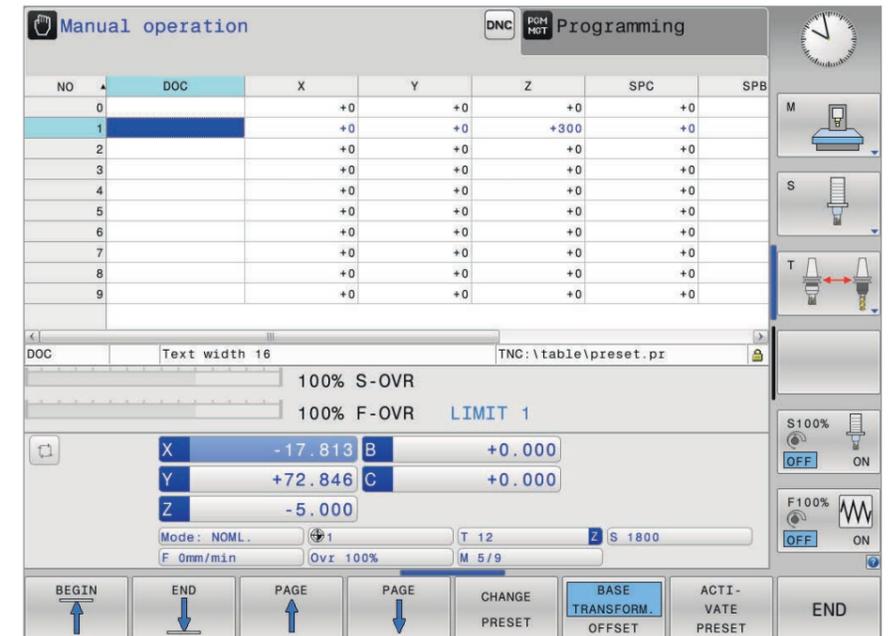
The TNC 320 manages and measures

The requirements gap between machining centers and typical tool and moldmaking machines is becoming ever narrower. Of course, the TNC 320 is also very capable of controlling automated manufacturing processes. It features the necessary functionality to manage and measure tools, as well as to inspect the machining process. It helps you reduce non-machining 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. This requires inserting a touch probe from HEIDENHAIN (see page 30) into the spindle in place of the tool. The resulting functionality allows you to do the following:
- Recognize a workpiece and call the appropriate part program
- Check whether all machining operations were conducted correctly
- Determine infeeds for finishing
- Detect and compensate for tool wear
- Ascertain the machining trend
- Log measured data
- Check whether tolerances were maintained, and configure various reactions, such as an error message
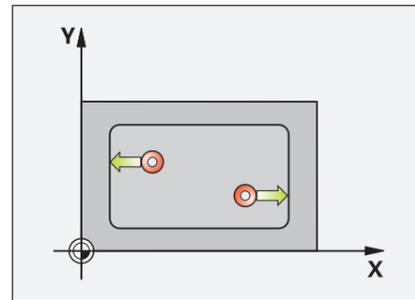
Measuring length
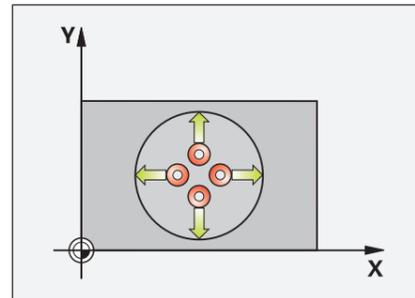
Measuring a circular pocket/hole
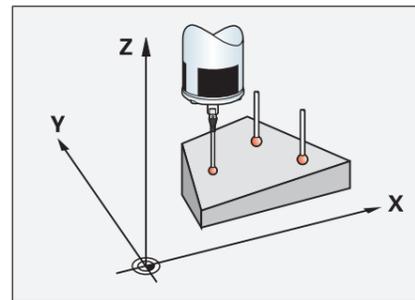
Measuring the angle of a plane

Tool measurement and automatic compensation of tool data

In conjunction with the TT touch probes (see page 31), the TNC 320 can automatically measure tools while they are in the machine. The TNC 320 then stores the determined values, such as tool length and radius, in its tool memory. In-process tool measurement allows you to immediately identify wear or breakage, thus avoiding scrap and rework. If the measured deviations lie outside the specified tolerances or if the monitored tool life has been exceeded, the TNC 320 locks the tool and automatically inserts a replacement tool.
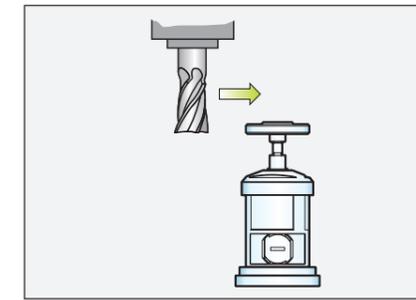
Measuring the tool radius
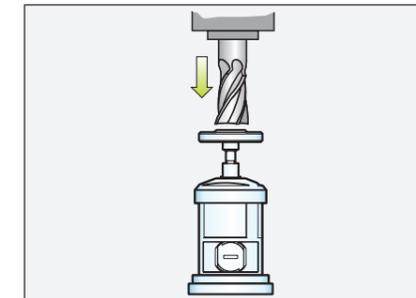
Measuring the tool length
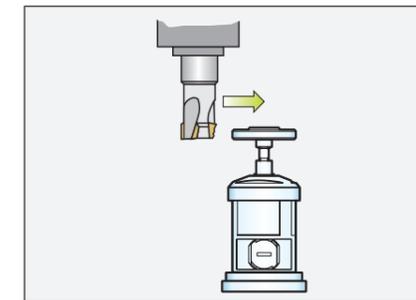
Measuring tool wear

Tool management

For machining centers with an automatic tool changer, the TNC 320 offers a central tool management function for any number of tools. This tool management function is a freely configurable file that can be optimally adapted 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 tool changes.

With the Extended Tool Management software option, you can also graphically prepare and display any data.*

*The control must be adapted for this function by the machine manufacturer.

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

Machine any contour slot with trochoidal milling

Many five-axis operations that initially appear to be highly complex can be broken down into common 2D movements that have been tilted about one or more rotary axes or wrapped around a cylindrical surface. In order for you to create and edit such programs quickly and easily without a CAD/CAM system, the TNC assists you with helpful functions.

Tilting the working plane*
Programs for contours and holes on inclined surfaces are usually very complex, requiring significant computing resources and programming work. But the TNC 320 can save you 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 with respect to the working plane about one or more rotary axes.

The PLANE function makes it easy for you to define a tilted working plane: tilted working planes can be specified in seven different ways depending on the information available in the workpiece drawing. Help images support you during program entry.

You can also use the PLANE function to define the positioning behavior during tilting to avoid unpleasant surprises when the program is run. The settings for defining the positioning behavior are identical for all PLANE functions, making operation much easier.

Cylinder surface machining*
The programming of contours made up of straight lines and circles on cylindrical surfaces for rotary tables is no problem with the TNC 320. You simply program the contour in the plane of an unrolled cylinder surface. The TNC 320 then performs the machining operation on the cylindrical surface of the cylinder.

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 control must be adapted for this function by the machine manufacturer.
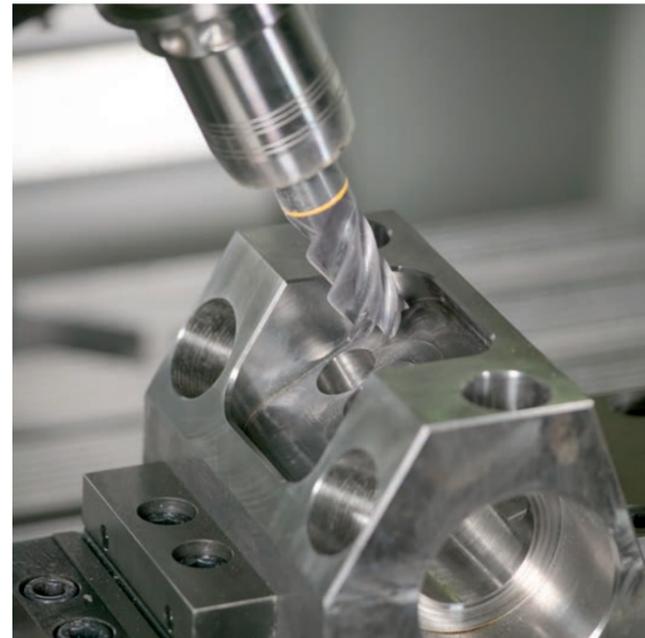
The ability to machine any slot at high efficiency from start to finish is the key benefit of trochoidal milling. In this cycle, roughing is performed with circular movements, onto which a forward linear movement is superimposed. This procedure is referred to as trochoidal milling. It is used particularly for milling high-strength or hardened materials, where the high loads placed on the tool and machine usually only permit small infeeds.

Thanks to the special cutting motion used in trochoidal milling, however, there are no added wear-inducing influences on the tool, and higher infeeds are possible. During hobbing, the entire cutting edge can be used as well. You thereby attain a higher material removal rate per tooth. Circular-motion plunging into the material exerts 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 can then be easily removed with a subsequent finishing cut.

The main benefits are as follows:
- Contact over entire cutter length
- Higher material removal rates
- Less strain on the machine
- Less vibration
- Integrated finishing of the side wall
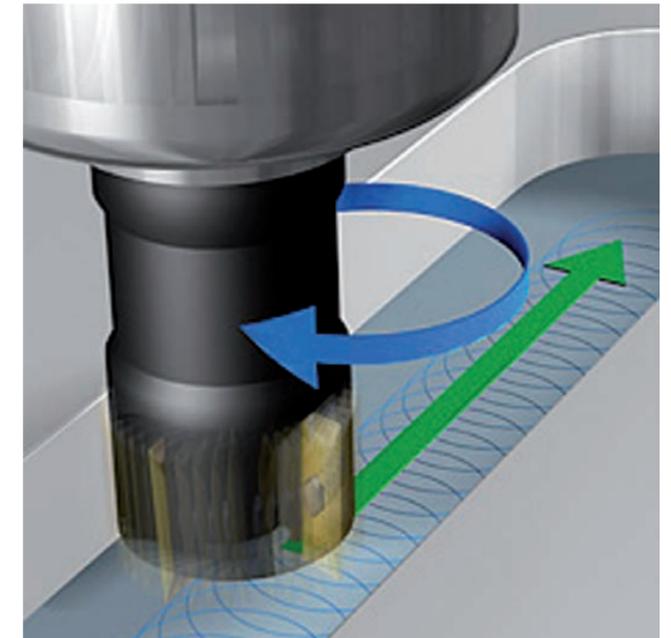
Besides being universally deployable, the TNC 320 is equally flexible when it comes to programming and machining.

Positioning with Manual Data Input
You don't need to create a complete program in order to start machining with the TNC 320. You can machine a workpiece step by step using both manual machining and automatic positioning in any sequence.

Programming at the machine
HEIDENHAIN controls are designed for shop-friendly programming right at the machine. Instead of requiring you to know G codes, the Klartext conversational programming language provides its own buttons and soft keys for programming straight lines, arcs, and cycles. At the push of a button, HEIDENHAIN Klartext dialog guidance opens and actively helps you during programming on the TNC. Unambiguous questions and prompts help you enter all the required information.

And even if you prefer ISO programming, the TNC 320 is still the right control: it displays soft-key rows dedicated to the most important ISO commands so that you can enter ISO programs directly. Or you connect a USB keyboard and use it to write the program.

All interface texts, including Klartext tips, dialog prompting, program steps, and soft keys, are available in numerous languages.

Creating programs offline
The TNC 320 is also well equipped to handle offline programming. Its interfaces allow the control to be integrated into networks in order to connect with programming stations or other data storage media. The TNC 320 can also run programs that were written in ISO format.
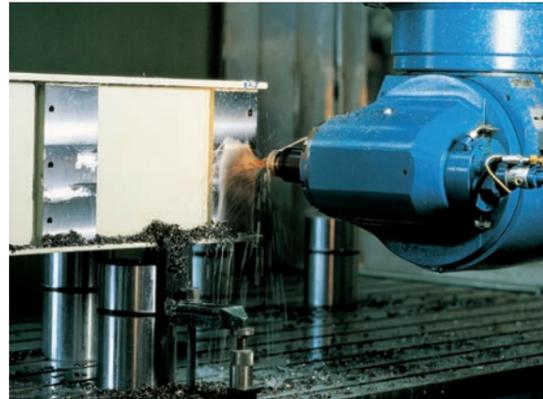
Programming graphics
The two-dimensional programming graphics give you additional assurance: while you are programming, the TNC 320 renders every programmed movement on the screen. You can opt for a plan view, side view, or front view. Tool paths or rapid-traverse movements can be hidden, and the image's scale can be changed.

Program verification graphics
For added certainty prior to machining, the TNC 320 can simulate and render the machining of the workpiece at high resolution. This simulation can be viewed in different formats:
• Plan view at different levels of depth
• Three projections
• 3D view

The 3D view lets you display the programmed tool-center path in three dimensions, and a powerful zoom function lets you see even the tiniest details. The 3D line graphics are especially helpful in verifying programs created offline, allowing you to avoid undesired machining marks on the workpiece, such as those that occur when the postprocessor outputs points incorrectly. In 3D view, the TNC also includes a measuring function that displays the coordinates of wherever your mouse happens to be within the graphics.

Program-run graphics
The program-run graphics display the workpiece in real time in order to show you the machining progress. Direct observation of the workpiece is usually impossible due to coolant and the safety enclosure. During machining, you can switch between various operating modes at any time in order to create programs, for example. And at the press of a button, you can cast a glance at the machining progress while programming.
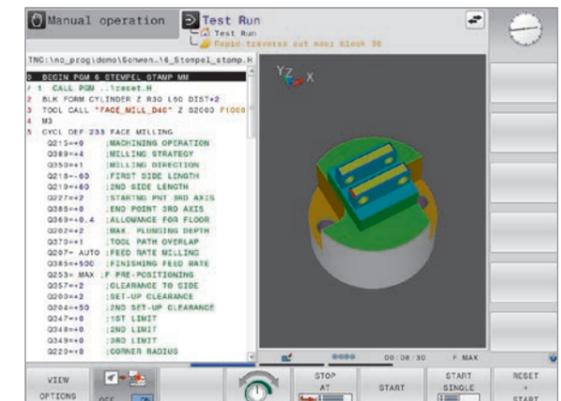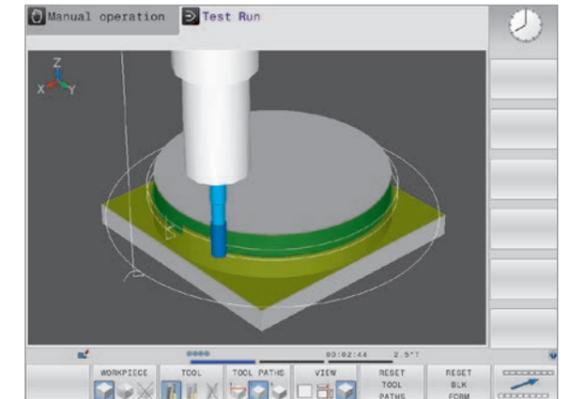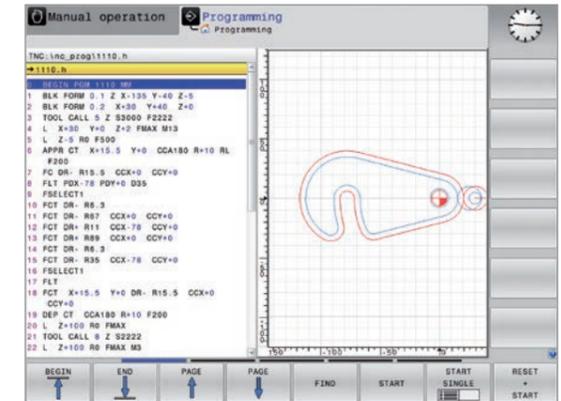
Programming 2D contours

Two-dimensional contours are the bread and butter of a modern machine shop, which is why the TNC 320 offers a variety of ways to handle them.

Programming with function keys

Are the contours properly dimensioned for NC machining? In other words, are the end points of the contour elements provided in either Cartesian or polar coordinates? If yes, then you can create the NC program straightaway using function keys.

Straight lines and circular elements

To program a line segment, for example, simply press the linear traverse key. The Klartext conversational programming language of the TNC 320 then prompts you for all of the information needed by the programming block, including the target coordinates, feed rate, tool compensation value, and machine functions. Function keys for circular movements, chamfers, and corner rounding also simplify your programming work. In order to avoid surface blemishes during approach or departure from the contour, movement must be seamless—that is, tangential.

You simply specify the starting point or end point of the contour and the approach or departure radius of the tool. The control handles the rest.

The TNC 320 can foresee a contour with radius compensation by up to 99 blocks, thus taking back-cutting into account and avoiding contour gouging. This could be the case, for example, when roughing a contour with a large tool.

FK free contour programming

Not all workpieces are properly dimensioned in accordance with the ISO standard. Thanks to FK, the control's programming function for free contours, you can simply enter the known data without needing to convert or calculate anything! In such cases, individual contour elements can remain undefined as long as the overall contour itself is defined. If the data allow for more than one mathematical solution, then the TNC 320 programming graphics helpfully render the possible variants to choose from.
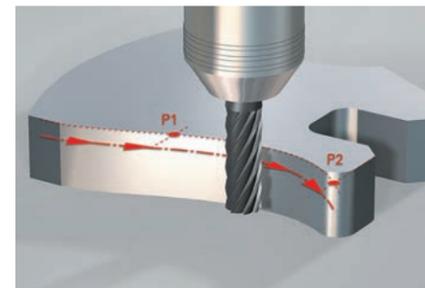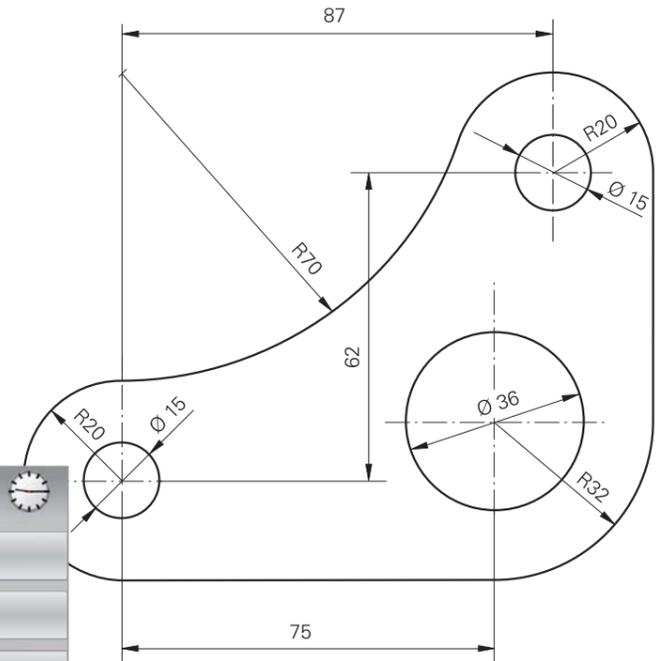
Circular path with seamless (tangential) transition from the preceding contour element, defined by its end point

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

Circular path defined by its radius, end point, and rotational direction
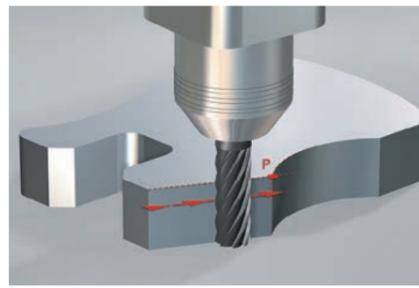
Straight line defined by its end point
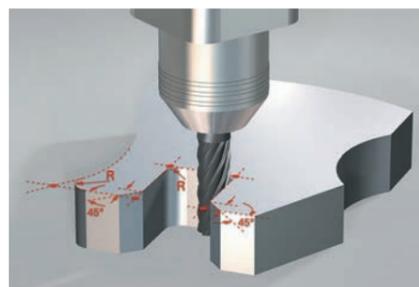
Corner rounding: circular path with a seamless (tangential) transition, defined by its radius and corner point

Chamfer defined by the corner point and chamfer length
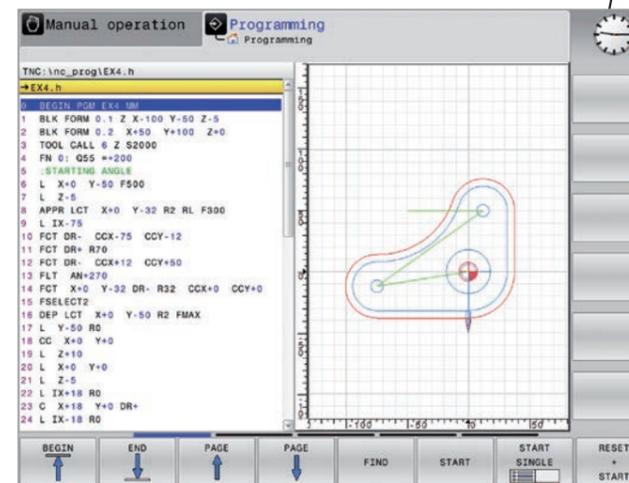
Practical cycles for recurring operations

Extensive machining cycles for milling and drilling

Frequently recurring operations consisting of multiple machining steps are stored as cycles in the TNC 320. Dialog guidance and help graphics illustrating the required input parameters support you in programming them.

Standard cycles

In addition to drilling and tapping cycles (with or without a floating tap holder), a variety of selectable cycles are available:
- Thread milling and reaming
- Engraving and boring
- Hole patterns
- Milling cycles for face milling a flat surface
- Roughing and finishing of pockets, slots, and studs

Cycles for complex contours

The Subcontour List (SL) cycles are particularly helpful for clearing pockets with combined contours. These cycles include machining cycles for pilot drilling, roughing, and finishing where the contour or subcontours are specified in subprograms. As a result, a single contour description can be used for various operations with different tools.

Up to twelve subcontours can be superimposed for the machining operation. The control automatically calculates the resulting contour and the tool paths for roughing or clearing the surfaces. Subcontours can be pockets or islands. Multiple pockets are combined to form a single pocket, with the tool bypassing any islands.

During roughing, a finishing allowance on the wall and floor surfaces is also taken into account by the TNC 320. When roughing with different tools, the control identifies residual material to be removed by smaller tools. A separate cycle is used for finishing to the final dimensions.

OEM cycles

As original equipment manufacturers (OEMs), machine manufacturers can contribute their special manufacturing know-how by designing additional fixed cycles and saving them in the TNC 320. The end user can write his own cycles as well. HEIDENHAIN makes this possible with its PC software CycleDesign, which allows you to create input parameters and arrange the TNC 320's soft key structure as desired.

Machining pattern

Positions to be machined are often arranged in patterns on the workpiece. The TNC 320's graphical support helps you program a wide range of machining patterns with ease and exceptional flexibility. You can define as many point patterns as desired with various numbers of points. During machining, you can then have all of the points executed as a unit, or each point individually.

Fast and easy programming of a scannable Data Matrix code

Cycle 224 (Data Matrix code pattern) lets you convert any text into a Data Matrix code and machine it onto your workpiece as a point pattern using, say, drilling operations. The Data Matrix code can be decoded with common scanning devices. Serial numbers and production data can thus be permanently machined onto the part. All you need to do after defining a machining cycle is enter the text (up to 255 characters) into the cycle and specify the size and position of the drilling pattern. The control automatically calculates the Data Matrix code and performs the machining work on its own.

3D machining with parametric programming

Simple, mathematically easy-to-describe 3D geometries can be programmed by means of parametric programming. At your disposal are basic arithmetical operations, trigonometric functions, roots, powers, logarithmic functions, parentheses, and logical comparisons with conditional jump instructions. Parametric programming also gives you a simple way to implement 3D operations. Of course, parametric programming is also well-suited for 2D contours that cannot be described with lines or circular arcs but rather through mathematical functions.

Reusing programmed contour elements
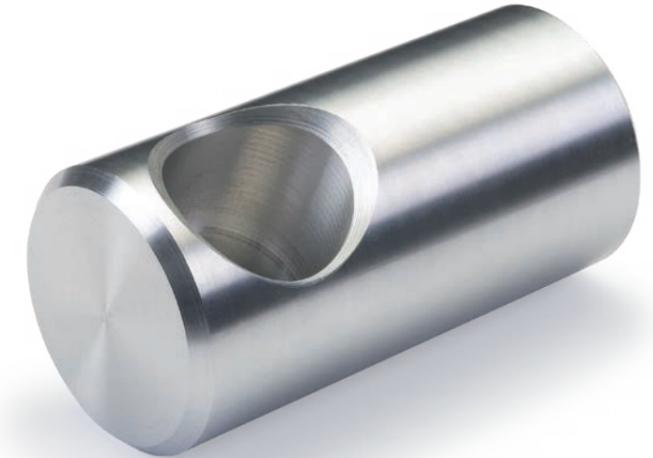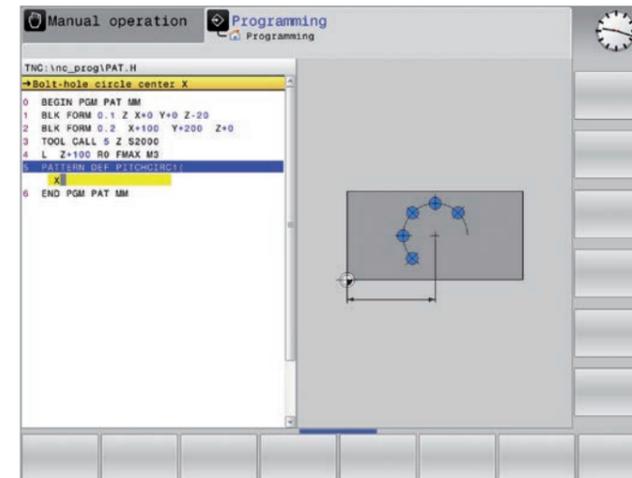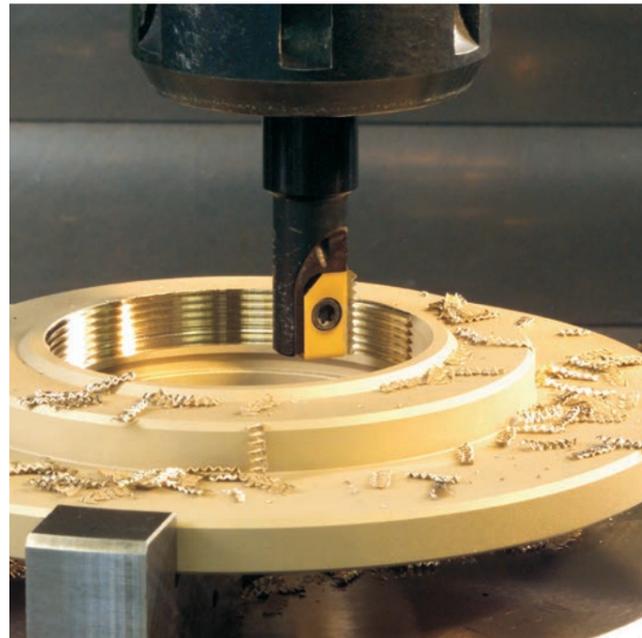
Coordinate transformations

If you need a pre-programmed contour at different locations and in different sizes on the workpiece, then the TNC 320 offers you a convenient solution in the form of coordinate transformations. 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 need to program it again. With its subprogram technology, the TNC can save you a great deal of programming time.

In program-section repeats, you can label a section of the program, and the TNC will repeat this section 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 location in your current program. This allows you to leverage already programmed and frequently required working steps or contours.

Of course you can also combine these programming techniques.

Do you have questions about a programming step, but your User's Manual is not at hand? No problem: both the TNC 320 control and the TNC 320 programming station feature TNCguide, a convenient help system that displays the user documentation in a separate window.

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

TNCguide generally displays the information instantaneously and in the correct context (context-sensitive help), immediately giving you the information you currently need. This function is particularly helpful for soft keys and provides a detailed explanation of their effects.

You can download the documentation in the desired language free of charge from the HEIDENHAIN website and save it to the appropriate language directory of the TNC hard disk.

The following user's manuals are available in the help system:
• Klartext Programming
• Setup, Testing, and Running NC Programs
• Cycle Programming
• ISO Programming
• TNC 320 Programming Station (installed only with the programming station)
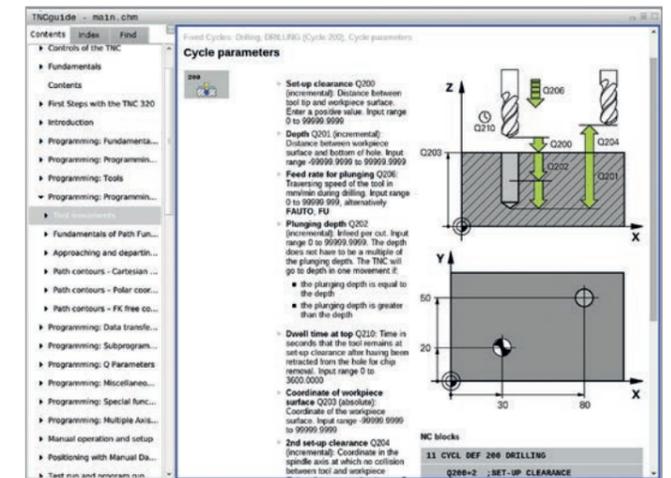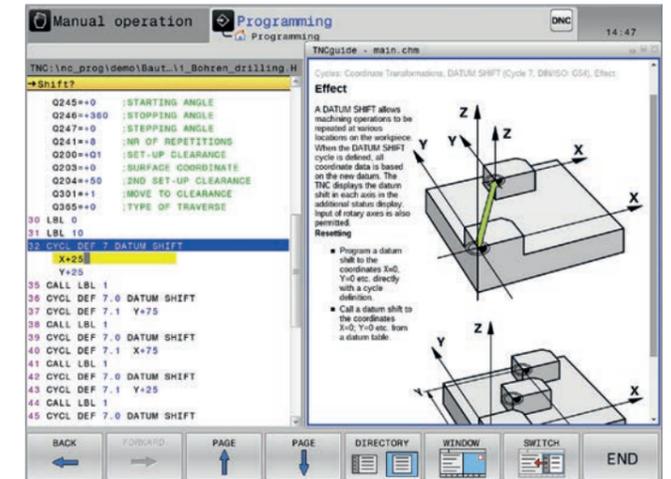
dynamic + efficiency

Efficient machining strategies are a key basis for economical NC manufacturing. Roughing processes, in particular, offer great potential for optimization because they usually make up a sizable portion of the total machining time.

Process reliability and maximum removal rates require optimal adaptation of the cutting data to the characteristics of the tool and workpiece material. Optimized Contour Milling (OCM) gives you a cutting data calculator that draws on a comprehensive integral materials database. You can adapt the automatically calculated cutting values with regard to the mechanical and thermal load on the tool. Tool service life can be managed with process reliability even at the highest possible removal rates.

With OCM, you can rough out any pocket and island with high process reliability and lower tool wear thanks to highly consistent process conditions. You simply program the contour as usual directly in Klartext or make use of the convenient CAD Import function. The control then automatically calculates the complex movements required for maintaining consistent process conditions. OCM takes void areas into account, which significantly reduces machining times (with NC software version 16 or later).

Advantages of OCM over conventional machining:
• Reduced thermal load on the tool
• Superior chip removal
• Consistent tool contact conditions (higher cutting parameters and higher material removal rate)

OCM is an effective, reliable, and convenient way to improve your throughput
• Create programs on the shop floor for any pocket or island
• Significantly higher machining speed
• Considerable reduction in tool wear
• More chips in less time

The OCM software option provides practical cycles for roughing, side-wall finishing, and floor finishing.

OCM also permits chamfering and deburring of contours. With this capability, only those areas that can be machined without collisions due to the tool geometry are machined.

Conventional machining	
S5000, F1200, a_p: 5.5 mm	
Overlap factor: 5 mm	
Machining time: 21 min 35 s	

Tool: VHM end mill Ø 10 mm
Workpiece material: 1.4104

Machining with OCM	
S8000, F4800, a_p: 22 mm	
Overlap factor: 1.4 mm	
Machining time: 6 min 59 s	

Tool: VHM end mill Ø 10 mm
Workpiece material: 1.4104

In order to machine standard shapes, OCM offers various figures that are then used as a pocket, island, or boundary for face milling in conjunction with other OCM cycles.

In the following machining example, both the machining time and tool wear were reduced **by a factor of three**.
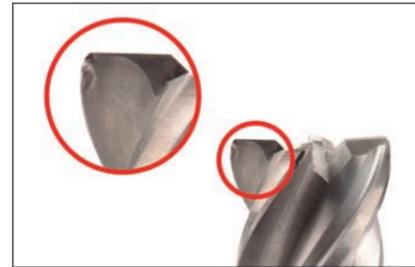
Machining without OCM: Tool after two parts
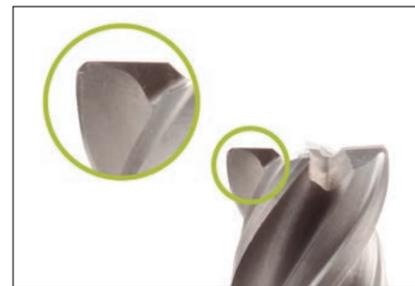
Machining with OCM: Tool after six parts
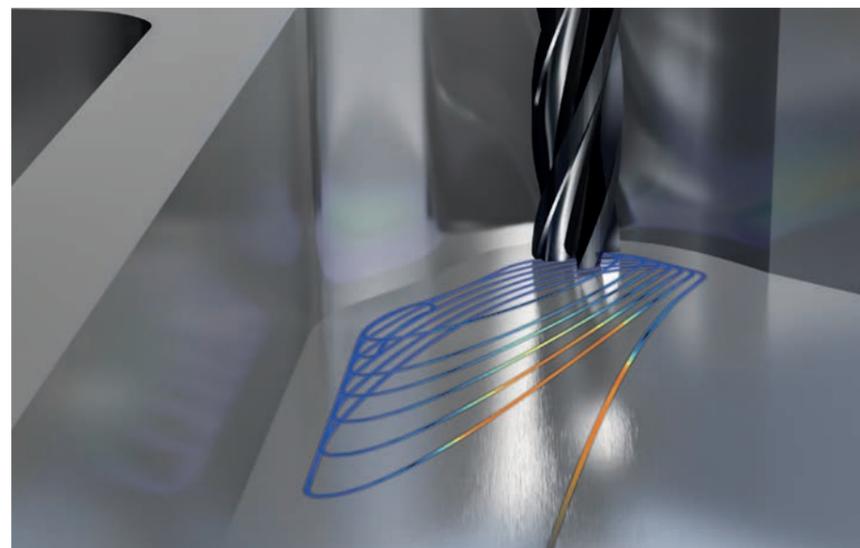
CAD Viewer
With the standardly available CAD Viewer software, you can open 3D CAD models and drawings directly on the TNC 320. Various view options as well as rotation and zoom capabilities allow for a detailed visual inspection and analysis of your CAD data. You can also use the viewer to ascertain position values from a 3D model. You simply select an arbitrary reference point in your drawing and select the desired contour elements. CAD Viewer then displays the coordinates of the elements in a window. CAD Viewer can depict the following file formats:
• STEP files (.STP and .STEP)
• IGES files (.IGS and .IGES)
• DXF files (.DXF)
• STL files

CAD Import (option)
Why program complex contours if you already have the drawing as a DXF, STEP, or IGES file? After all, you have the ability to extract contours or machining positions from these types of CAD files. This not only saves time otherwise spent on programming and testing, but you can also be sure that the finished contour is exactly in accordance with the design engineer's specifications.

Extracting machining information directly from CAD data offers additional possibilities, particularly for the creation of NC programs with a tilted machining plane. You can also define the preset with a 3D basic rotation of the 3D model and set a datum with the appropriate 3D rotation in 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. In the defined working plane, you can extract contours and machining positions, and apply them to the NC program.

Selecting contours is very convenient: you can select any element by clicking it with the mouse. Once you select a second element, the TNC 320 detects your desired machining direction and initiates automatic contour detection. The TNC 320 automatically selects all clearly identifiable contour elements until the contour closes or branches out. This method lets you define elaborate contours with just a few clicks of the mouse. You can then easily copy the selected contour via the clipboard to an existing Klartext program.

You can also select machining positions and save them as point files, especially in order to apply hole positions or starting points for pocket machining. This can be done very easily by selecting the desired area with the mouse. In a pop-up window with a filter function, the TNC 320 displays all of the hole diameters within the area you have selected. To select the desired hole diameters and restrict the number of hole positions, click the corresponding filter symbol to change the filter limits. A zoom function and various configuration possibilities round out the functionality of CAD Import.

It is also possible to define the contour program's resolution for using it in older TNC controls. You can also define a transition tolerance if the contour elements are not completely connected.

The following positions can be defined 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, including in their extension
- Intersection of a line and a circular arc
- Intersection of a line and a circle

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

Generating STL files (option)
The CAD Model Optimizer software option makes it possible to generate STL files from 3D models. The TNC 320 covers the 3D model displayed in CAD Viewer with a mesh of triangles. This simplifies the original model and removes errors, such as small holes in a solid or self-intersections of a surface. The TNC 320 then generates an STL file that you can use for various functions of the control. For example, you can then easily repair faulty files of fixtures or tool holders.

The networked TNC 320
You can integrate the TNC 320 into networks and connect it to PCs, programming stations, and other data storage media. Even the basic version of the TNC 320 is equipped with a latest-generation Fast Ethernet data interface in addition to its RS-232-C/V.24 data interface. The TNC 320 uses the TCP/IP protocol to communicate with NFS servers and Windows networks without requiring additional software. The fast data transfer at rates of up to 1000 Mbit/s ensures very short transmission times.

The transmitted programs are saved in the TNC 320's internal memory 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.
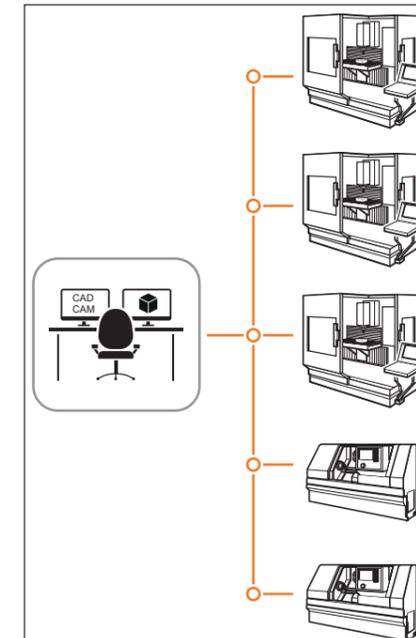
Programs for data transfer
With the aid of the free PC software TNCremo from HEIDENHAIN, you can do the following:
- Transfer remotely stored part programs and tool or pallet tables in both directions
- Make backups

With the powerful TNCremoPlus PC software, you can also transfer the screen contents from the control to your PC using the live-screen function.
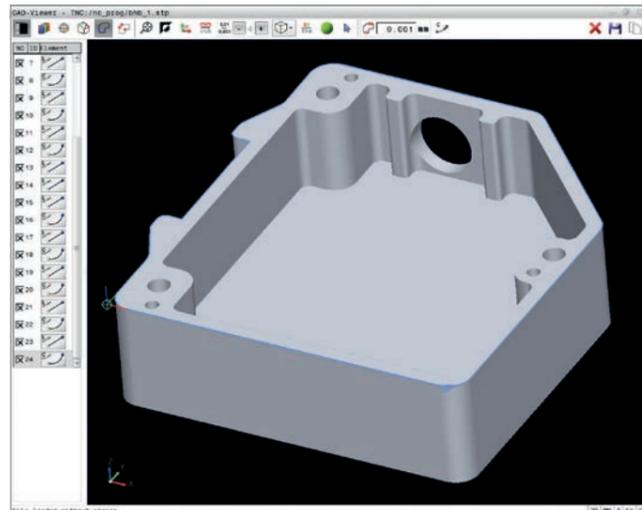
Contour selection from an imported CAD file
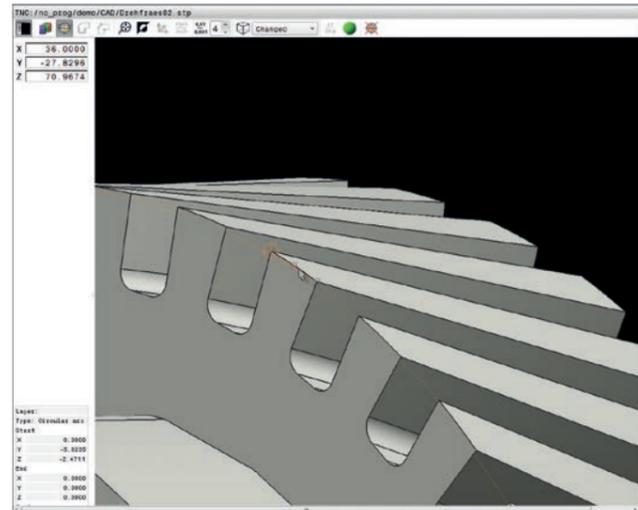
Display of a 3D model in CAD Viewer

PDF viewer

The integrated PDF viewer enables you to open PDF files directly on the control. This enables you to easily view work instructions, drawings, or other information on the TNC.

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

You can also open the following further file formats right on the TNC using the appropriate programs, and even edit some of them:
• Text files with the extensions .txt or .ini
• Graphic files with the extensions .gif, .bmp, .jpg or .png
• Table files with the extensions .xls, .xlsx or .csv
• HTML files

An operating panel with integrated touchpad or an external USB pointing device is required for operation.
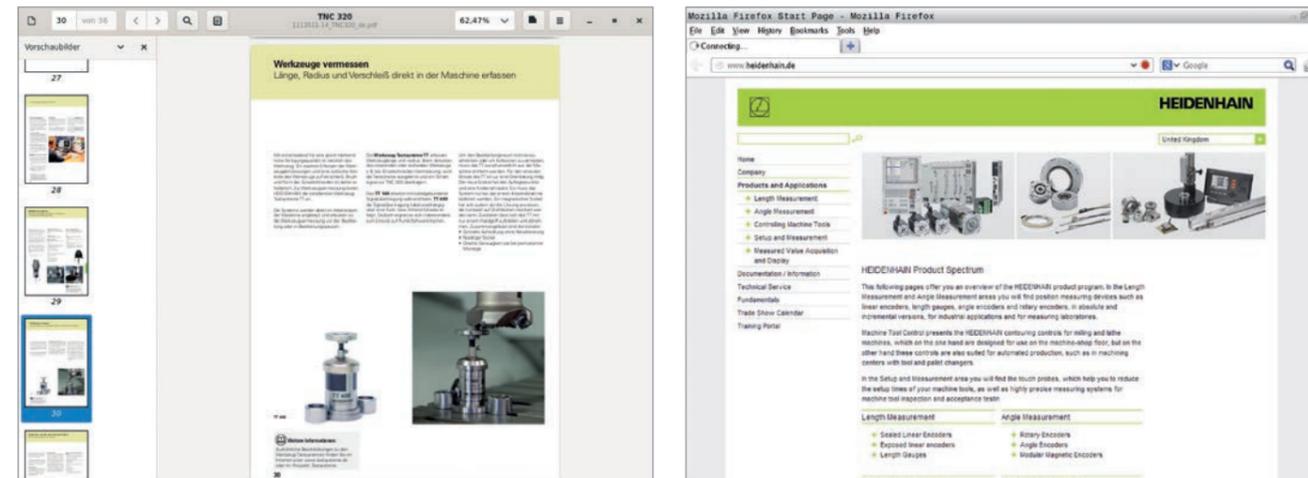
Why a programming station?

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

The programming, testing, and optimizing of HEIDENHAIN Klartext or G-code programs for the TNC 320 with the programming station substantially reduces machine idle times. And it's easy to get used to because every keystroke feels the same: the programming station's keyboard is just like the one on the machine.

Testing 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 reliably 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 control, it is ideally suited for apprentice- and advanced-level training. Programming and Test Run modes work exactly as they would on the machine. This gives the trainee the confidence needed to operate a real machine.

By being programmable in either Klartext or ISO, the TNC 320 programming station is also well suited for TNC programming training in school settings.

Your workstation

The programming station software runs on a PC. The PC screen shows you the same TNC user interface as the control and offers the familiar graphical support. Depending on the version of the programming station, there are multiple ways of using it.

The free demo version contains all of the TNC 320's functions and allows short programs to be saved. Programming is performed on the PC keyboard.

The version with a TNC operating panel lets you create programs in the accustomed manner on a keyboard featuring the same function keys as the control on the machine. It also has a PC keyboard for ISO programming, file names, and comments.

You can also work without the TNC operating panel: a virtual keyboard for operating the programming station is shown on the PC screen. This virtual keyboard provides the TNC 320's most important dialog-opening 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.com/programming-stations*

Workpiece measurement
Setup, preset setting and measuring with touch trigger probes

Tool measurement
Measuring length, radius and wear inside the machine

Workpiece touch probes* from HEIDENHAIN help you keep costs down in the shop and in serial production: when used in conjunction with the TNC 320's probing cycles, they enable the automated execution of setup, measurement, and inspection functions.

Upon contact with the workpiece surface, the stylus of a TS touch trigger probe is deflected. In that instant, the TS generates a triggering signal that, depending on the model, is transmitted to the control either by cable or by radio or infrared transmission.

The touch probes are inserted directly into the machine tool spindle and can be equipped with various taper shanks depending on the machine. The ball tips, which are made of ruby, are available in different diameters with various stylus lengths.

∗ The machine and TNC must be specially adapted by the machine manufacturer.
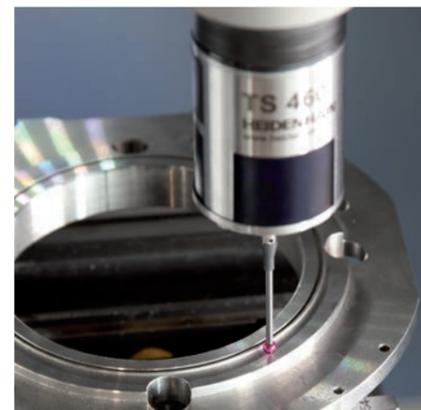
Cable-bound touch probes
For machines requiring manual tool changes, as well as for grinding machines and lathes:
TS 260
- Axial or radial cable connection
- High probing accuracy

Wireless touch probes
For machines with an automatic tool changer:
TS 460
- Standard touch probe for radio and infrared transmission
- Compact dimensions
- Energy-saving mode
- Optional collision protection
- Thermal decoupling

TS 642
- Activation via switch in taper shank
- Infrared transmission

TS 760
- High scanning accuracy
- High reproducibility
- Low probing forces
- Radio and infrared transmission

Transceiver
Radio or infrared transmission is established between the TS or TT touch probe and the SE transceiver:
SE 660
- For radio or infrared transmission (hybrid technology)
- Same SE for TS 460 and TT 460

SE 661
- For radio or infrared transmission (hybrid technology)
- Same SE for TS 460 and TT 460
- EnDat functionality for transmission of the switching state, diagnostics, and additional information

A key factor in obtaining consistently high production quality is, of course, the tool itself. This requires exact measurement of the tool dimensions and periodic inspection of the tool for breakage, wear and tooth shape. For tool measurement, HEIDENHAIN offers the TT triggering tool touch probes.

These touch probes are installed directly within the machine's workspace, where they enable 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 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/tilting tables.

To prevent limiting the machining space and to avoid collisions, the TT touch probe must occasionally be removed from the machine. The touch probe's new magnetic base features three contact points and a setting screw. As a result, the touch probe requires calibration only during initial installation and can then be re-installed and removed with a quick hand movement.

Here are the benefits in a nutshell:
- Fast re-installation without recalibration
- Low-profile base
- Same accuracy as a permanent installation

SE 660

TS 460

TS 460 with collision protection

TT 460

Further information:
You can find detailed descriptions of workpiece touch probes on the Internet at *www.heidenhain.com* or in the *Touch Probes for Machine Tools* brochure.

Further information:
You can find detailed descriptions of tool touch probes on the Internet at *www.heidenhain.com* or in the *Touch Probes for Machine Tools* brochure.

Positioning with an electronic handwheel
Fine motion control of axes

You can set up the workpiece by manually jogging the axes with the axis direction keys, but this task is easier and more finely controllable with the electronic handwheels from HEIDENHAIN.

In this method, the axis slides are moved by the feed motor in accordance with the handwheel's rotation. For particularly high sensitivity, you can incrementally set the amount of traversing distance for each handwheel rotation.

Panel-mounted handwheel
The HR 130 panel-mounted handwheel from HEIDENHAIN can be integrated into the machine operating panel or mounted to a different part of the machine.

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 workspace. The axis keys and certain function keys are integrated into the housing. This lets you set up the machine or switch between the axes you want to move from wherever you happen to be standing with the handwheel. 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, simply attach it to the machine using its built-in magnets.

Expanded range of functions of the HR 520 and HR 550
- Definable traversing distance per revolution
- Display for operating mode, actual position value, programmed feed rate, spindle speed, and error messages
- Override potentiometers for feed rate and spindle speed
- Handwheel offset
- Selection of axes by means of keys and 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 manufacturer

HR 550

Overview
User functions

User functions	Standard	Option	
Short description	✓	0 1	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 format ISO programming via soft keys or USB keyboard
Position feedback	✓ ✓ ✓		Nominal position 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	✓ ✓		With respect to the path of the tool center With respect to the cutting edge
Parallel operation	✓		Program creation with graphical support while another program is running
Rotary table machining		8 8	Programming of cylindrical contours as if in two axes Feed rate in mm/min
Contour elements	✓ ✓ ✓ ✓ ✓ ✓ ✓		Straight line Chamfer Circular path Circle center Circle radius Tangentially connecting circular arc Corner rounding
Contour approach and departure	✓ ✓		Via straight line: tangential or perpendicular Via circular arc
FK free contour programming	✓		FK free contour programming in HEIDENHAIN Klartext format with graphical support for workpiece drawings not dimensioned for NC
Program jumps	✓ ✓ ✓		Subprograms Program-section repeat Any program as a subprogram

User functions	Standard	Option	
Fixed cycles	✓		Cycles for drilling, peck drilling, 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
	✓		Multi-operation machining of straight and circular slots
	✓		Multi-operation machining of rectangular and circular pockets
	✓		Circular and linear point patterns
	✓		Contour train, contour-parallel contour pocket
	✓		OEM cycles (special cycles developed by the machine manufacturer) can be integrated
	✓		Engraving cycle: engrave text or numbers in a straight line or on an arc
	✓		Point pattern: Data Matrix code
		167	Optimized Contour Milling (OCM) cycles: optimization of roughing processes
Coordinate transformations	✓		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, In, 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 320
	✓		Graphical support for the programming of cycles
	✓		Comment and structure blocks in the NC program
Teach-In	✓		Actual positions can be transferred directly to the NC program
Program verification graphics Display modes	✓		Graphical simulation of the machining operation, even while another program is being executed
	✓		Plan view / projection in three planes / 3D view, also in tilted working plane / 3D line graphics
	✓		Detail enlargement
Programming graphics	✓		Rendering (2D graphics) of entered NC blocks in Programming and Editing mode, even while another program is being executed
Program-run graphics Display modes	✓		Real-time graphical simulation during execution of the program
	✓		Plan view / view in three planes / 3D view
Machining time	✓		Calculation of machining time in the Test Run mode of operation
	✓		Display of the current machining time in the Program Run operating modes

User functions	Standard	Option	
Returning to the contour	✓		Mid-program startup at any block in the program and movement to the calculated nominal position for continuation of the machining operation
	✓		Program interruption, contour departure and return
Preset management	✓		For storing any presets
Datum tables	✓		Multiple datum tables for storing workpiece-specific datums
Touch probe cycles	✓		Touch probe calibration
	✓		Compensation of workpiece misalignment
	✓		Manual or automatic preset setting
	✓		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 parallel axes included in the position display of the associated principal axis (sum display)
	✓		Defining the principal and secondary axes in the NC program enables execution 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

Software options

Option number	Option	With NC software 771851- or later	ID	Comment
0 1	Control Loop Qty.	01	354540-01 353904-01	Additional control loop
8	Adv. Function Set 1	01	536164-01	**Rotary table machining** • Programming of cylindrical contours as if in two axes • Feed rate in mm/min
		01		**Interpolation** • Circular in three 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 of contours from 3D models
93	Ext. Tool Management	02	676938-01	**Extended tool management** • Tooling list (list of all tools of the NC program) • T usage sequence (sequence of all tools inserted during the program)
133	Remote Desk. Manager	01	894423-01	Display and remote operation of external computer units (e.g., a Windows PC)
152	CAD Model Optimizer	16	1353918-01	Conversion and optimization of CAD models • Fixtures • Workpiece blank • Finished part
167	Opt. Contour Milling	16	1289547-01	OCM: optimize roughing processes and fully utilize milling tools with the integrated cutting data calculator

Accessories

Accessories	
Electronic handwheels	• **HR 510** portable handwheel • **HR 520** portable handwheel • **HR 550** portable wireless handwheel • **HR 130** panel-mounted handwheel
Workpiece measurement	• **TS 260** touch trigger probe with cable connection, or • **TS 460 / TS 460** touch trigger probe with radio or infrared transmission, or • **TS 642** workpiece touch probe with infrared transmission
Tool measurement	• **TT 160** 3D touch trigger probe, or • **TT 460** 3D touch trigger probe with radio or infrared transmission
Software for PCs	• **RemoteAccess** for remote diagnostics, monitoring, and operation • **CycleDesign** for creating your own cycle structure • **TNCremo** for data transfer—free of charge • **TNCremoPlus** for data transfer with live-screen function
Programming station	Control software for PCs for programming, archiving, and training • Single-station license with original control operating panel • Single-station license with operation via virtual keyboard • Network license with operation via virtual keyboard • Demo version (operated with virtual keyboard or PC keyboard—free of charge)

Specifications

Specifications	
Components	• Main computer with TNC keyboard and integrated 15-inch screen with soft keys
Operating system	• HEROS 5 real-time operating system for machine control
Memory	• SSDR: 240 GB • CFAST: 30 GB
Input resolution and display step	• Linear axes: down to 0.1 µm • Rotary axes: down to 0.0001°
Interpolation	• Linear in four axes • Circular in two axes • Circular in three axes with tilted working plane • Helical: superimposition of circular and straight paths
Block processing time	• 6 ms (3D straight line without radius compensation)
Axis feedback control	• Position resolution: signal period of the position encoder/1024 • Cycle time of position controller: 200 µs
Error compensation	• Linear and nonlinear axis error, backlash, reversal peaks during circular movements, thermal expansion • Static friction, reversal error, sliding friction
Data interfaces	• Network interface for remote operation of the TNC using the TNCremo or TNCremoPlus software from HEIDENHAIN • 2 x Ethernet interface, 1 Gigabit • USB 3.0 (and one USB 2.0 on the operating panel); the number of ports depends on the hardware used
Diagnostics	• Fast and simple troubleshooting through integrated diagnostic aids
Ambient temperature	• Operation: +5 °C to +40 °C • Storage: −20 °C to +60 °C

HEIDENHAIN
Mastering nanometer accuracy

HEIDENHAIN
worldwide