



TNC 620

Compact Contouring
Control for Milling, Drilling,
and Boring Machines

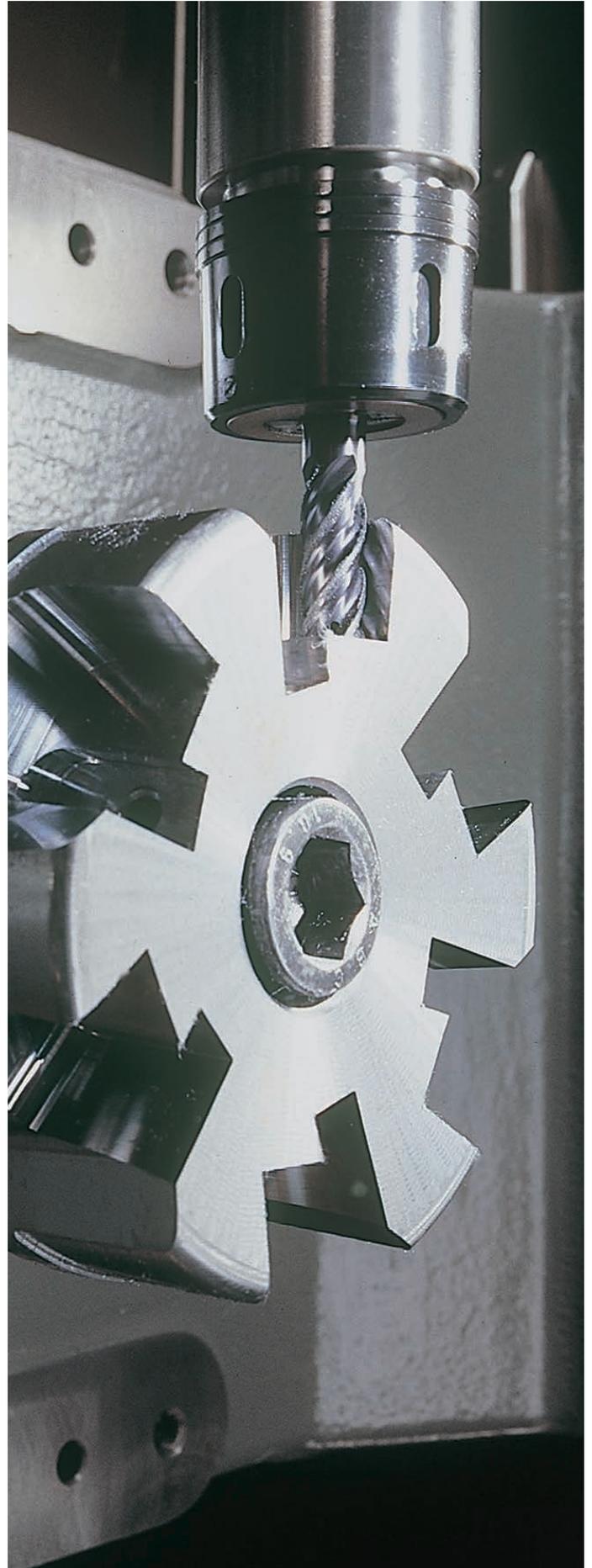




**TNC 620
with touchscreen**



**TNC 620
with operating keys and
alphanumeric keyboard**



The functions and specifications described in this brochure apply to the TNC 620 with NC SW 81760x-07.

Contents

The TNC 620...

Where can it be used?	Compact and versatile The right control for milling, drilling, and boring machines	4
What does it look like?	Intuitive and user-friendly State-of-the-art multitouch operation Highly practical user interface	6
What can it do?	Fast and reliable machining at high contour fidelity Optimal tool guidance by the TNC 620	10
	Machining with five axes TNC-controlled swivel head and rotary table	12
	Minimize setup times The TNC 620 makes setup easy	14
	Automated machining The TNC 620 measures, manages, and communicates Pallet management and serial machining	16
	How is it programmed?	Programming, editing, and testing A full range of possibilities with the TNC 620 Graphical support in any scenario
	Programming in the shop Straightforward function keys for complex contours Programing Data Matrix codes and free contours Practical cycles for recurring operations Reusing programmed contour elements Rapid availability of all information	20
	Intelligent machining Dynamic Precision Machine any contour slot with trochoidal milling Active Chatter Control option (ACC) Optimized roughing processes with OCM (option)	26
	The TNC 620 programming station	31
	Open to outside information Processing CAD files with the TNC 620 Uniformly digital job management with Connected Machining StateMonitor: collect and evaluate machine data	32
What accessories are there?	Positioning with an electronic handwheel Fine motion control of axes	37
	Workpiece measurement Setup, preset setting, and measuring with touch trigger probes	38
	Tool measurement Measuring length, radius, and wear inside the machine	39
	Inspecting and optimizing machine accuracy Easy calibration of rotary axes with KinematicsOpt (option)	40
	... At a glance	Overview Specifications, user functions, accessories, options, comparison of controls

Compact and versatile

The right control for milling, drilling, and boring machines

For more than 40 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 machine operator.

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

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

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

Multitouch operation

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

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



Shop-oriented programming

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

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

Offline program creation

The TNC 620 is easy to program remotely as well. Its Fast Ethernet interface ensures extremely short upload times, even for long programs. In addition, data can be transferred quickly and easily via the USB interface.

CAD viewer as standard

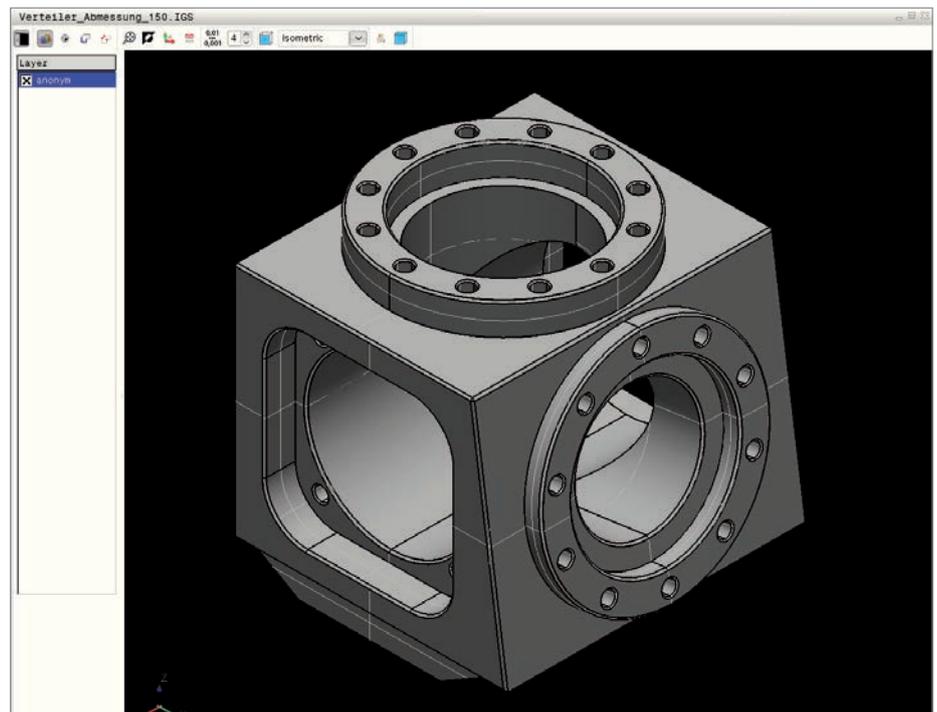
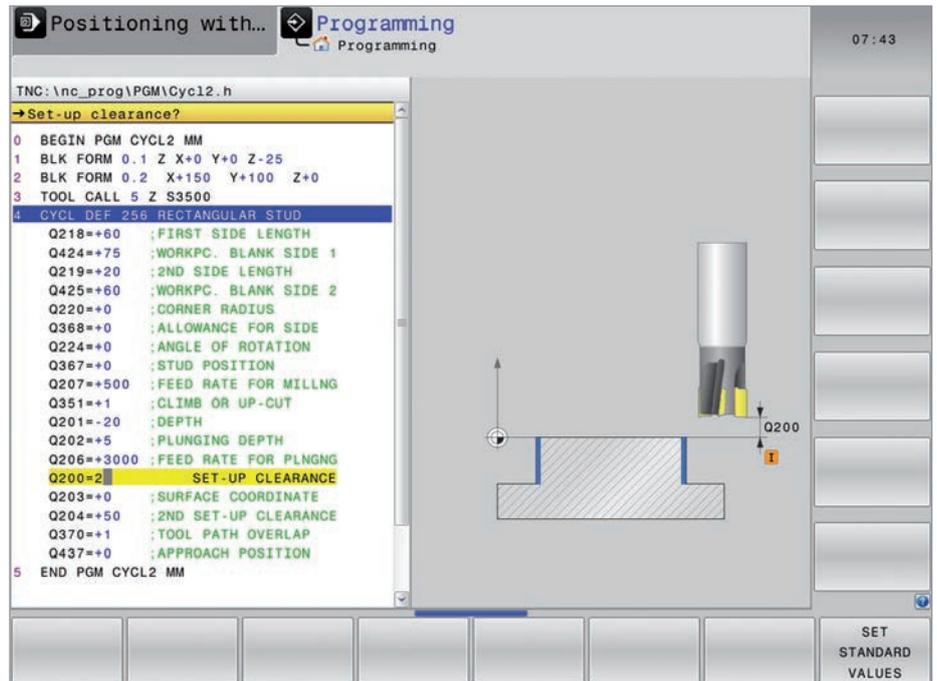
Display your CAD data in the .step or .iges format directly on the TNC 620. The standard integrated CAD viewer can open many common 3-D formats and display them on the TNC 620 screen.

Finely detailed graphics

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

Advanced Dynamic Prediction (ADP)

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



Intuitive and user-friendly

State-of-the-art multitouch operation

The screen

In a clear format, the large TFT color flat-panel displays all of the information you need for programming, operating, and checking the control and the machine, including 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.

While the program is running, status displays are available that inform you about the tool position, current program, coordinate transformations, and more. The TNC 620 even shows the current machining time.

Well-thought-out touchscreen

The conventional operating approach of the TNC with its easy-to-read control panel has proven itself over many years, with operators around the world using the TNC's dialog keys, navigation keys, and soft keys. But the touchscreen version of the TNC 620 now supports you with a particularly innovative and user-friendly method of operation. This approach combines the proven benefits of the HEIDENHAIN controls with a new method that uses tapping, swiping, and dragging gestures.

Easy to operate

The TNC 620 can be conveniently operated with a connected mouse, but many machine shops lack an adequate surface for a mouse. A touchscreen, however, eliminates the need for additional workspace for input devices and makes everyday operation of the TNC 620 even easier thanks to swiping, direct selection of operating elements, and menu navigation. The touchscreen is particularly helpful when using the CAD Import function, allowing you to rapidly zoom, drag, or select with simple gestures.



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

PLC functions keys (soft keys) for machine functions

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

Axis keys, numeric keypad, and navigation

Shortcut menu

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

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

Practical touchscreen

The touchscreen was conceived for harsh shop environments. It is splash-proof, scratch-resistant, and certified for IP54 protection. To clean your screen, simply select the "Touchscreen Cleaning" mode. This locks the screen to prevent unintended operation.

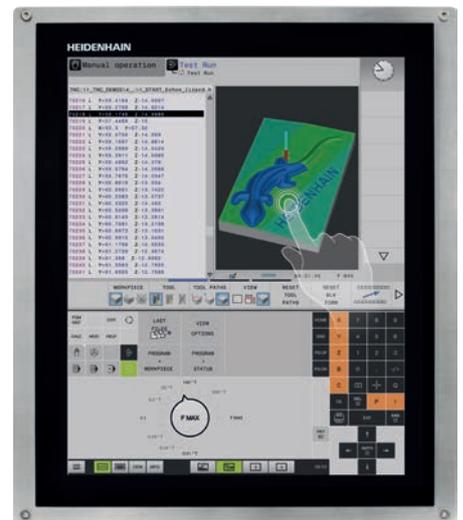
Optimal screen display

The user interface of the TNC 620 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 620 displays them in color-coded categories. A color-coded warning triangle is also displayed.

Gestures for multitouch operation

The screen of the TNC 620 can be operated with gestures such as those commonly used on mobile devices. For example, you can use two fingers to zoom in and out, and swiping lets you quickly navigate through the soft key bar, programs, and menus.

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

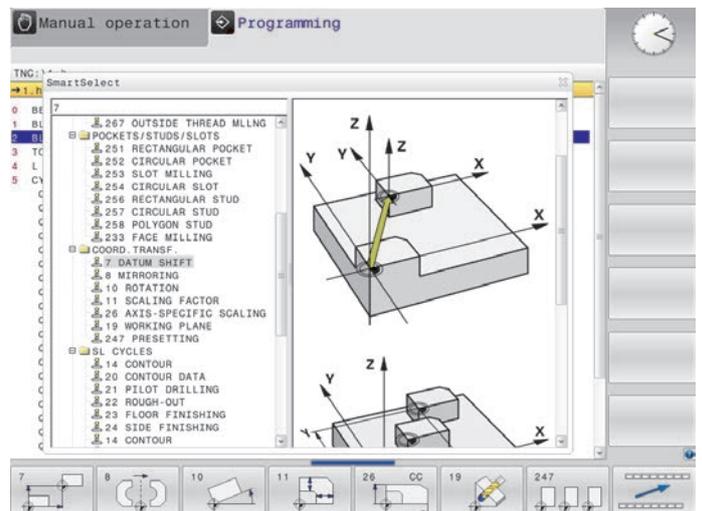
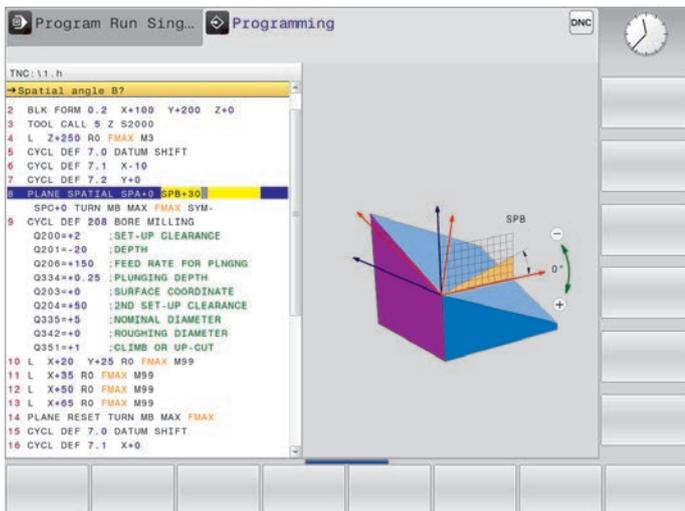


Highly practical user interface

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 620 offers a number of features that make working with the control even simpler and easier than before.

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, 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 comprehensive: line number, program function, input values, comment. To make sure you always keep an overview even in complex programs, the TNC 620 displays the individual program elements in different colors. The color syntax highlighting improves your overview when editing NC programs. This allows you, for example, to see at a glance where the editable input values are.

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

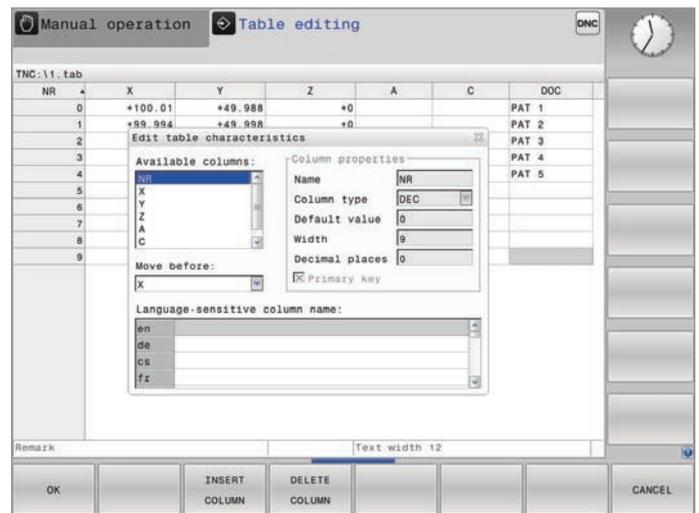
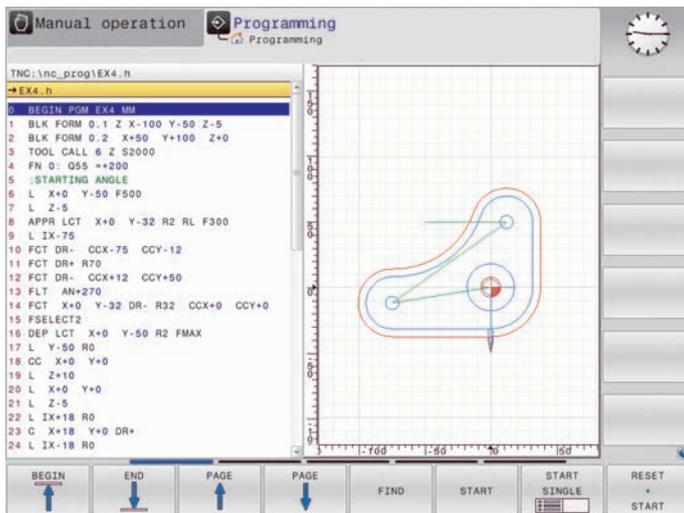
In the info line, the TNC 620 helps you orient yourself by displaying the respective status of the submode. The function is comparable to the history function in web browsers.

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 620 lets you define various roles and access rights for users, ensuring that each user can act only within his assigned rights. This prevents the unintentional or unauthorized deletion of files or content from system files. In addition, many functions are available only with the proper rights. Consequently, the user administration feature of the TNC 620 not only provides greater data security but also increases machine operating safety.



Fast and reliable machining at high contour fidelity

Optimal tool guidance by the TNC 620

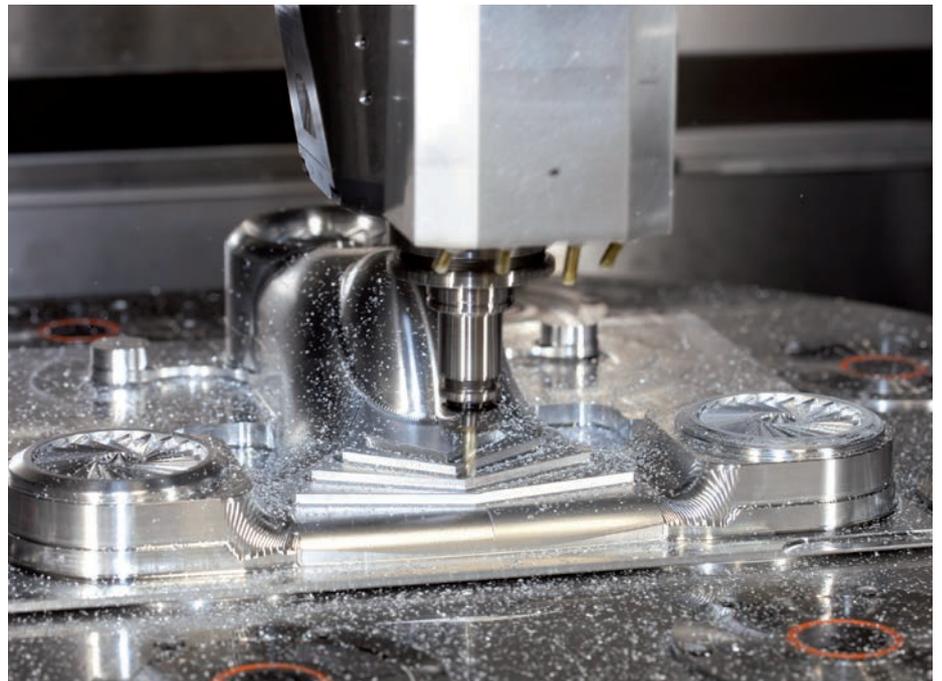
Very high contour fidelity and surface quality

TNC controls from HEIDENHAIN are known for their **low-jerk and speed/acceleration-optimized motion control**, enabling optimized surface quality and workpiece accuracy. With the TNC 620, you are able to leverage the latest technological developments. The TNC 620 anticipates and thinks along with you, dynamically calculating the contour ahead of time. In addition, special filters suppress specific machine-specific natural frequencies.

With its **Look Ahead** function, the TNC 620 recognizes directional changes ahead of time and adapts the traversing speed to the shape of the contour and the surface to be machined. You simply program the maximum machining velocity as feed rate and, in **Cycle 32 TOLERANCE**, enter in the control the maximum permissible deviations from the ideal contour. The TNC 620 automatically adapts the machining to the tolerance that you define. This method prevents contour defects.

Advanced Dynamic Prediction (ADP)

ADP expands upon the previous look-ahead calculation of the permissible maximum feed rate profile. ADP compensates for differences in the feed rate profiles resulting from the point distribution on neighboring paths, especially in NC programs originating from CAM systems. Among other benefits, this results in particularly symmetrical feed rate behavior on the back-and-forth paths during bidirectional finish milling, as well as very smooth feed rate curves on neighboring milling paths.



Fast machining and computing processes

With a short block-processing time of at most 1.5 ms, the TNC 620 can run fast look-ahead calculations for optimal use of the machine's dynamic parameters. Functions such as ADP and Look Ahead therefore ensure not only high contour fidelity and surface quality but also optimize the machining time.

One of the reasons for the TNC 620's high speed is its **uniformly digital control design**. It consists, on the one hand, of the integrated digital drive technology from HEIDENHAIN, and on the other hand all control components are interconnected with digital interfaces—the control components via HSCI (HEIDENHAIN Serial Controller Interface), and the encoders via EnDat 2.2. As a result, exceptionally high feed rates can be implemented. In the process, the TNC 620 is capable of interpolating up to five axes at the same time. To attain the required cutting speeds, the TNC 620 can digitally control spindle speeds of up to **100 000 rpm**.

The TNC 620's powerful 5-axis machining capability even lets you economically manufacture complex 3-D contours. Programs for such contours are usually created offline with CAM systems and typically contain numerous short line blocks that are transferred to the control. With its short block-processing time, the TNC 620 quickly executes even complex NC programs. The control's computing power can handle complex look-ahead calculations in simpler NC programs as well. Thus, regardless of the data volume of the NC programs from your CAM system, the TNC 620 will machine the workpiece to near-perfect conformity with your program.



Machining with five axes

Swivel head and rotary table controlled by the TNC

Many five-axis operations that initially appear to be highly complex can be broken down into common 2-D 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 problems quickly and easily without a CAD/CAM system, the TNC supports you with helpful functions.

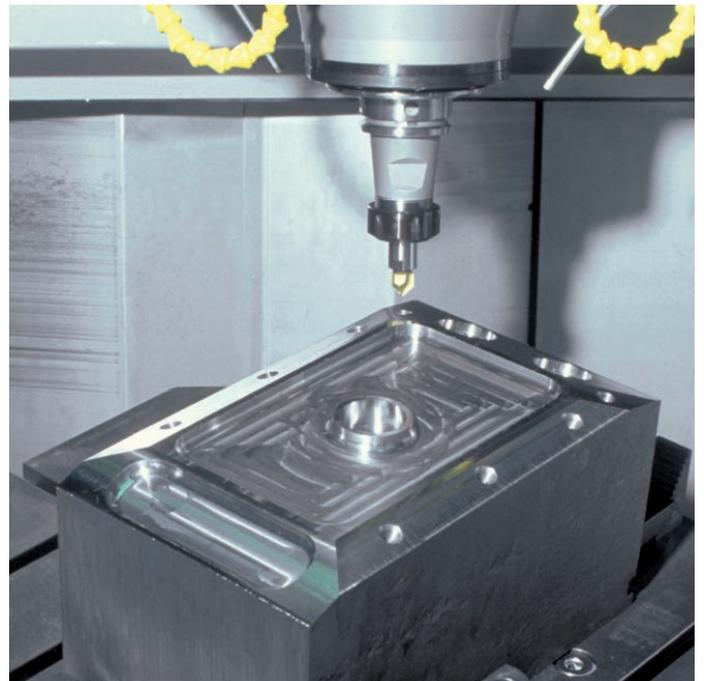
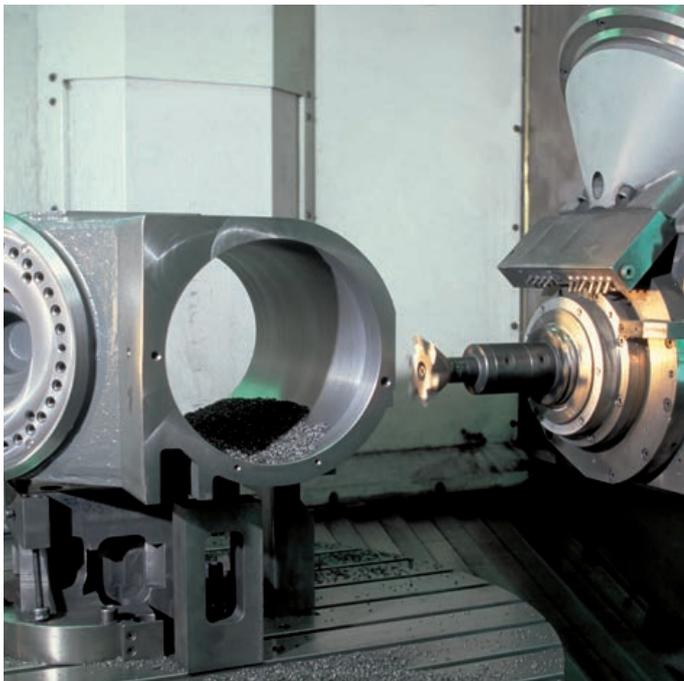
Tilting the working plane* (option 8)

Programs for contours and holes on inclined surfaces are often very complex and require time-consuming computing and programming work. But the TNC 620 can save you a great deal of programming time. You program the machining operation as usual in the main plane—for example, in X/Y. The machine then runs the program in a plane that has been tilted by one or more rotary axes with respect to the main plane.

The PLANE 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.

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



Cylinder surface machining* (option 8)

Programming contours made up of straight lines and circles on cylindrical surfaces for rotary tables is no problem with the TNC 620. You simply program the contour in the plane of an unrolled cylinder surface. The TNC 620 then performs the machining operation on the cylindrical surface of the cylinder

The TNC 620 features four cycles for cylindrical surface machining:

- Slot milling (the slot width is the same as the tool diameter)
- Guide-groove milling (the slot width is greater than the tool diameter)
- Ridge milling
- Outside contour milling

Manual axis motion in the tool direction on 5-axis machines

Tool retraction is very important in five-axis machining. The virtual tool axis function provides assistance by allowing you to move the tool in the direction of the tool axis using external direction keys or a handwheel. This function is especially useful if you want to

- retract the tool in the direction of the tool axis during interruption of a 5-axis machining program,
- use the handwheel or external direction keys to perform an operation in Manual mode with an inclined tool, or
- move the tool with the handwheel in the active tool axis direction during machining (option 21 required).

Feed rate for rotary tables in mm/min* (option 8)

By default, the feed rate of rotary axes is programmed in degrees/min. However, the TNC 620 can interpret this feed rate in mm/min as well. The feed rate at the contour is then independent of the distance of the tool center from the center of the rotary axis.

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



Minimize setup times

The TNC 620 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. This is 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.

The TNC 620 features application-oriented, real-world setup functions that support the user, help reduce non-productive time, and enable overnight, unattended production. In conjunction with the **touch probes**, the TNC 620 offers numerous probing cycles for automatic workpiece setup, setting the preset, and measuring workpieces and tools.

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 the electronic handwheels from HEIDENHAIN (see page 37). 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

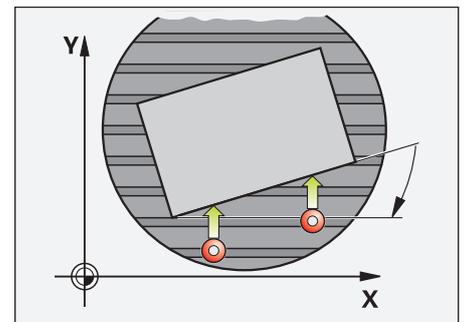
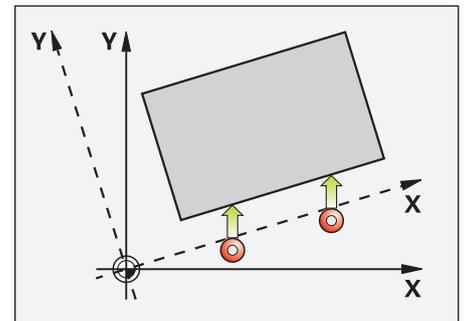
In many cases, the workpiece must be probed at unseen positions or in cramped spaces, and the standard probing feed rate is usually too fast for this. In such situations, you can use the override knob to change the feed rate during probing without affecting the accuracy.

Workpiece alignment (option 17)*

Touch probes from HEIDENHAIN (see page 38) and the probing functions of the TNC 620 keep you from having to perform time-consuming workpiece alignment:

- Clamp the workpiece in any position.
- The touch probe determines the actual workpiece position by probing a surface, two holes, or two studs.
- The TNC 620 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 620 offers manual, automatic, and semiautomatic cycles for compensating for misalignments in two or three dimensions

* Automatically enabled for HEIDENHAIN touch probes.



Compensating for workpiece misalignment

by rotating the coordinate system or turning the table

Setting presets

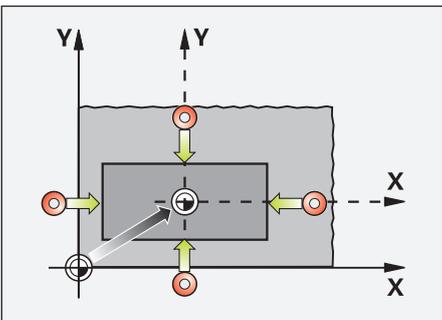
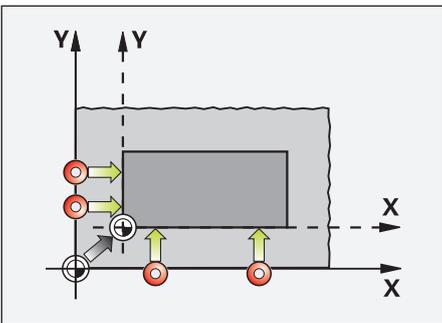
A preset allows you to assign a defined value displayed by the TNC to any position on the workpiece. Finding this point quickly and reliably reduces non-productive time and increases machining accuracy.

The TNC 620 features probing cycles (option 17) for automatic setting of presets. Once found, you can save these presets

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

Setting a preset

on a corner, for example, or in the center of a circular stud



Preset management with the preset table

Preset management enables flexible machining, shorter setup times, and higher throughput. This makes it much easier to set up the machine.

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 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

Datum tables let you save positions or measured values from the workpiece. Datums are always defined relative to the active preset.

The screenshot shows the 'Manual operation' screen with a 'Programming' mode. A table lists preset data for 9 lines. Below the table, the coordinate display shows X: -17.813, Y: +72.846, Z: -5.000. The screen also displays various machine parameters like S100%, F100%, and T 12.

NO	DOC	X	Y	Z	SPC	SPB
0		+0	+0	+0	+0	+0
1		+0	+0	+300	+0	+0
2		+0	+0	+0	+0	+0
3		+0	+0	+0	+0	+0
4		+0	+0	+0	+0	+0
5		+0	+0	+0	+0	+0
6		+0	+0	+0	+0	+0
7		+0	+0	+0	+0	+0
8		+0	+0	+0	+0	+0
9		+0	+0	+0	+0	+0

Coordinate Display:

X: -17.813 B: +0.000
 Y: +72.846 C: +0.000
 Z: -5.000

Mode: NOML. T 12 Z S 1800
 F 0mm/min Ovr 100% M 5/9

Buttons: BEGIN, END, PAGE, PAGE, CHANGE PRESET, BASE TRANSFORM OFFSET, ACTIVATE PRESET, END

Automated machining

The TNC 620 measures, manages, and communicates

The requirements gap between machining centers and typical tool and moldmaking machines is becoming ever narrower. Of course, the TNC 620 is capable of controlling automated manufacturing processes. It features the necessary functionality to initiate the proper machining operation in concatenated machining scenarios with individual workpieces in any workholding arrangement.

Tool management

For machining centers with automatic tool changers, the TNC 620 offers a central tool memory for any number of tools. This tool memory area is a freely configurable file that can be optimally adapted to your needs. You can even have the TNC 620 manage your tool names. The control prepares the next tool change while the current tool is still cutting. This significantly reduces the non-cutting time required for tool changes.

With the Expanded Tool Management option, the control shows only the required input fields regardless of the selected tool type. Any data can also be graphically depicted.*

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

Inspecting workpieces for proper machining and dimensional accuracy (option 17)

The TNC 620 features a number of measuring cycles for checking the geometry of the machined workpieces. This requires the insertion of a 3-D touch probe from HEIDENHAIN (see page 38) instead of the tool in the spindle. This enables you to

- recognize a workpiece and call the appropriate part program,
- check whether all machining operations were conducted correctly,
- determine infeeds for finishing,
- detect and compensate tool wear,
- check the workpiece geometry and sort the parts,
- log measured data,
- ascertain the machining error trend.

Tool measurement and automatic compensation of tool data (option 17)

In conjunction with the TT tool touch probe (see page 39), the TNC 620 lets you automatically measure tools in the machine. The TNC 620 then stores the determined values, such as tool length and radius, in its tool memory. By inspecting the tool during machining, you can quickly detect wear or breakage for preventing scrap and rework. If the measured deviations lie outside the specified tolerances or if the monitored tool life has been exceeded, the TNC 620 locks the tool and automatically inserts a replacement tool.



Pallet management and serial machining

Pallet Management (option 22)

Pallet management functionality lets you automatically machine workpieces in any order. Upon insertion of the pallet, the correct machining program and preset are automatically selected. You can also apply coordinate transformations and measuring cycles in the part programs.

Batch Process Manager (option 154)

Batch Process Manager is a powerful function for pallet machining and serial production. With its intuitive user interface, you can plan your production process and receive important information about upcoming machining operations.

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

Batch Process Manager displays the following information in advance:

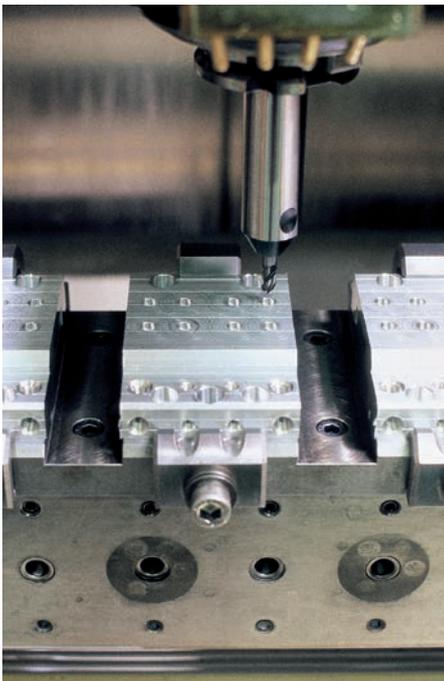
- The sequence of operations
- The time of next manual intervention
- The program duration and run time
- Status information about the preset, tool, and program

Tool-oriented machining (option 22)

In tool-oriented machining, a single machining step is performed on all of a pallet's workpieces before moving to the next machining step. This keeps the number of tool changes to a minimum and significantly shortens the machining time.

The TNC 620 supports you with convenient fillable forms that allow you to assign a tool-oriented operation to a pallet with multiple workpieces. As always, you still write the machining program in the normal workpiece-oriented sequence.

You can also use this function even if your machine does not support pallet management. In this case, you simply define the position of the workpiece on the machine table in the pallet file.



Program run, full sequence

TNC: \nc_prog\demo\Pallet\01_START_BPM.p

Necessary manual intervent...	Object	Time	Next manual intervention:
External tool	NC_SPOT_DRILL...	13:46	24m 6s
External tool	DRILL_D16	13:46	

Program	End	Preset	T	Pgm	Sts
Pallet: 1					
└ 1_Pris..rism.h	13:38	✓	✓	✓	⊞
└ 2_Haus..ouse.h	13:47	✓	✗	✓	⊞
Aufspan..ng: 123		⊞	⊞	?	
Palette: 2					

100% S-OVR

100% F-OVR LIMIT 1

X	+0.000	A	+0.000
Y	-25.000	C	+0.000
Z	+300.000		

Modus: ACTL. 7 T 3 Z S 5000

F 0mm/min Ovr 100% M 5/9

INSERT REMOVE MOVE RESET THE STATUS MACHINING METHOD EDIT OFF ON DETAILS OFF ON SELECT

Programming, editing, testing

A full range of possibilities with the TNC 620

Besides being universally deployable, the TNC 620 is equally flexible when it comes to programming and editing.

Positioning with Manual Data Input

You don't need to create a complete program in order to start machining with the TNC 620. You can machine a workpiece step by step using both manual machining and automatic positioning in any sequence.

Programming at the machine

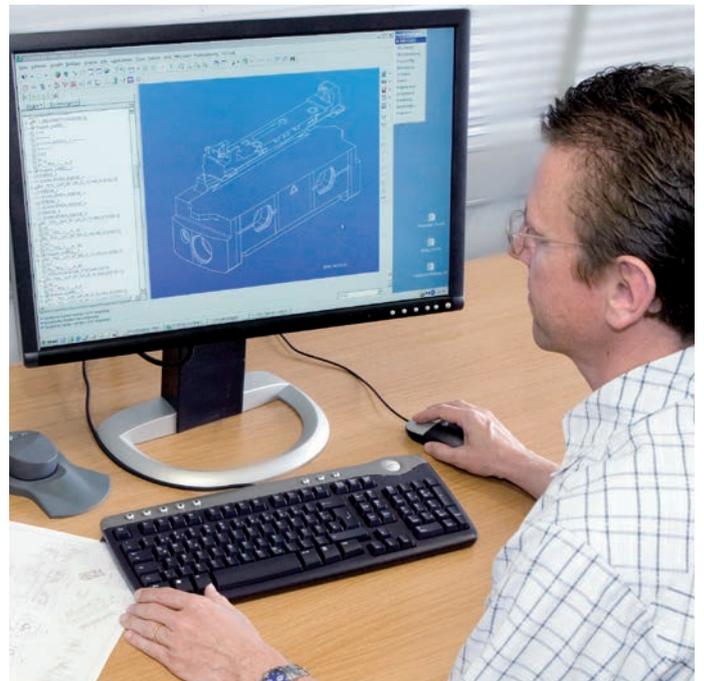
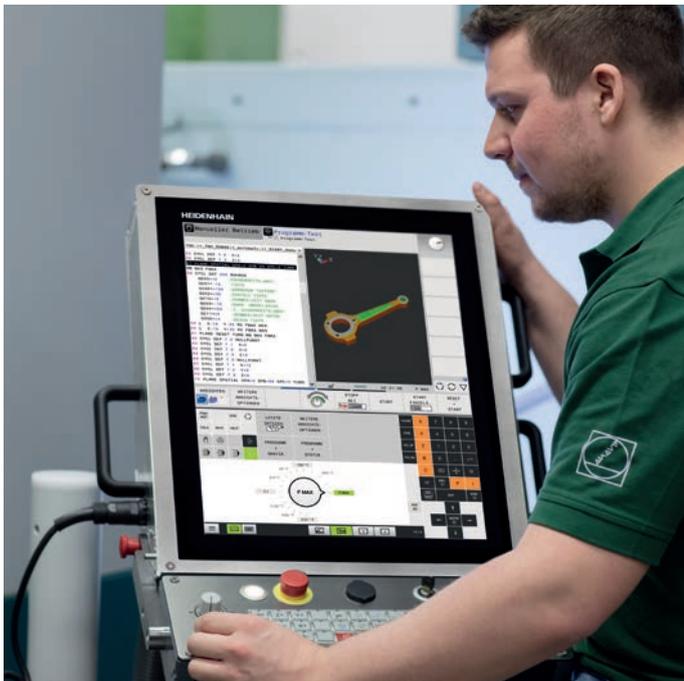
HEIDENHAIN controls are designed for shop-friendly programming right at the machine. Instead of requiring you to know G-codes, the Klartext conversational programming language provides its own buttons and soft keys for programming straight lines, arcs, and cycles. At the push of a button, HEIDENHAIN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

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

And even if you prefer **ISO programming**, the TNC is still the right control: you can enter ISO command letters via soft keys or directly on the alphanumeric keyboard.

Creating programs offline

The TNC 620 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 620 can also run programs that were written in ISO format.



Graphical support in any scenario

Programming graphics

The two-dimensional programming graphics give you additional assurance: while you are programming, the TNC 620 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 (option 20)

For added certainty prior to machining, the TNC 620 can simulate and render the machining of the workpiece at high resolution. This simulation can be viewed in different formats:

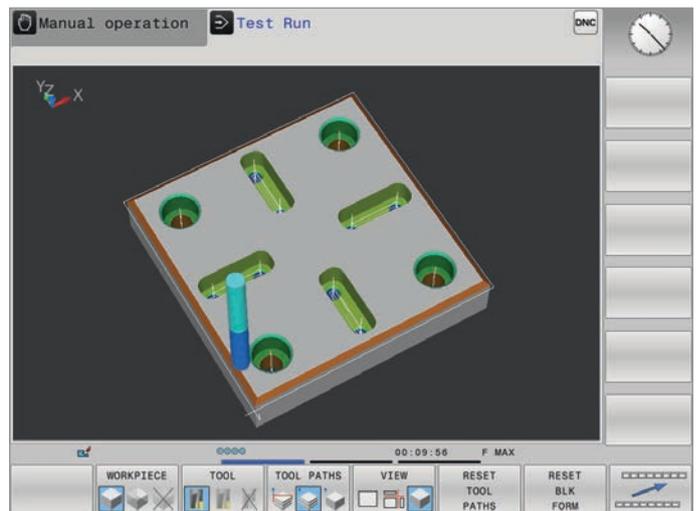
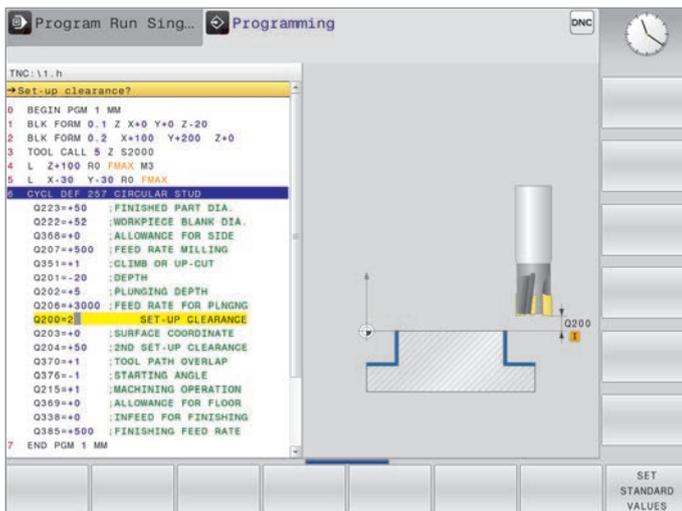
- Plan view at different levels of depth
- Three projections
- 3-D view

The type and quality of the image can be selected, and details can be displayed at magnification. The TNC 620 also displays the calculated machining time in hours, minutes, and seconds.

The 3-D 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 3-D 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. The 3-D view of the TNC also provides a measuring function. When you move the mouse pointer anywhere within the image, the coordinates of that position are displayed. The measurement window also displays tool information.

Program-run graphics (option 20)

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 in the shop

Straightforward function keys for complex contours

Programming 2-D contours

Two-dimensional contours are the bread and butter of a modern machine shop, which is why the TNC 620 offers a variety of ways to handle them.

Programming with function keys

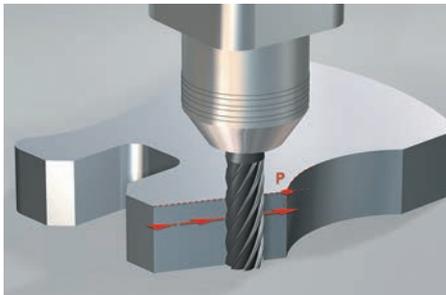
Are the contours properly dimensioned for NC machining? In other words, are the end points of the contour elements provided in either Cartesian or polar coordinates? If yes, then you can create the NC program straightaway using function keys.

Straight lines and circular elements

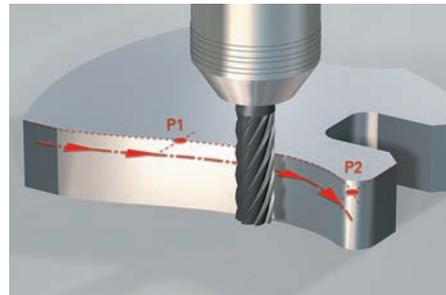
To program a line segment, for example, simply press the linear traverse key. The Klartext conversational programming language of the TNC 620 then prompts you for all of the information needed by the programming block, including the target coordinates, feed rate, tool compensation value, and machine 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.

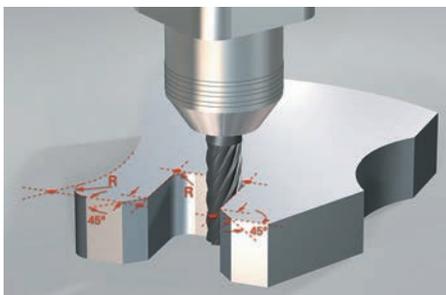
The TNC 620 can foresee a contour with radius compensation by up to 99 blocks (option 21), thus taking back-cutting into account and avoiding contour gouging. This could be the case, for example, when roughing a contour with a large tool.



Straight line defined by its end point



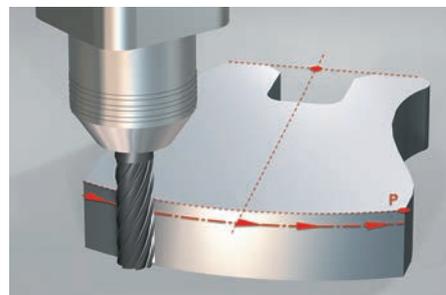
Circular path with seamless (tangential) transition from the preceding contour element, defined by its end point



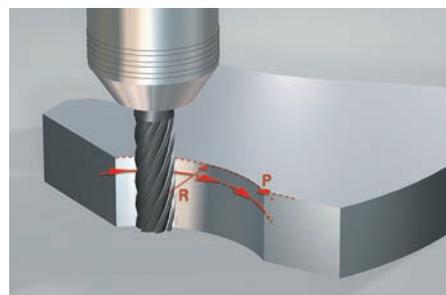
Corner rounding: circular path with smooth (tangential) connection on both sides, defined by radius and corner point



Chamfer defined by the corner point and chamfer length



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



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

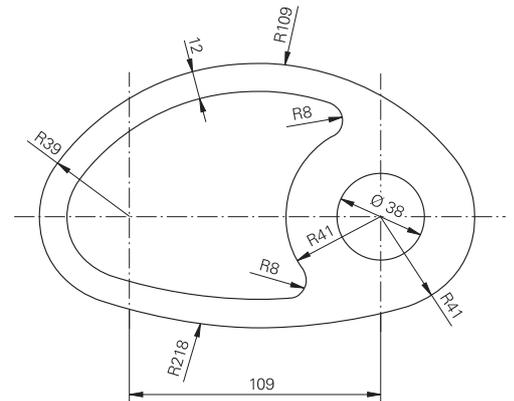
Programming Data Matrix codes and free contours

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.

FK free contour programming (option 19)

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 from the drawing yields more than one possible mathematical solution, then the helpful TNC 620 programming graphics show you the possible variants for your selection.



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 620. 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:

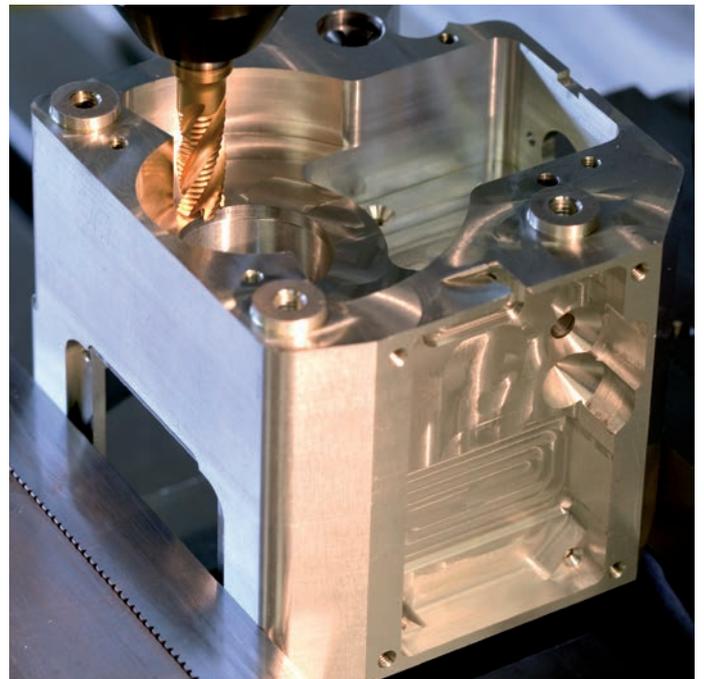
- Thread milling
- Reaming
- Engraving
- Boring
- Hole patterns
- Milling cycles for face milling a flat surface
- Roughing and finishing of pockets, slots, and studs

Cycles for complex contours (option 19)

Of great help in roughing pockets with any contour are the control's **SL cycles** (SL = Subcontour List) and **OCM** (Optimized Contour Milling) functionality. These functions refer to roughing and finishing cycles in which the contour or subcontours are defined in subprograms. Thus, a single contour description can be used for more than one operation 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 620. 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 (option 19)

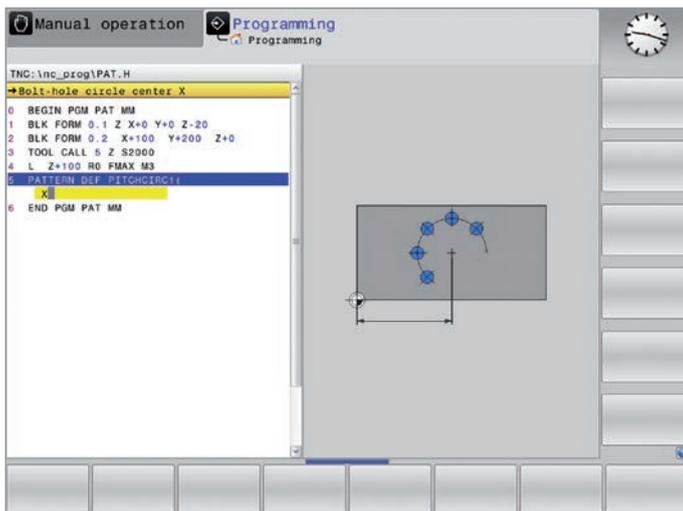
Machine manufacturers can incorporate their specialized production expertise in the form of additional machining cycles and store them in the TNC 620. The end user can write his own cycles as well. HEIDENHAIN makes this possible with its PC software CycleDesign, which allows you to create input parameters and arrange the TNC 620's soft key structure as desired.

Program machining patterns with ease and flexibility

Positions to be machined are often arranged in patterns on the workpiece. The TNC 620'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.

3-D machining with parametric programming

Simple, mathematically easy-to-describe 3-D 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 3-D operations. Of course, parametric programming is also well-suited for **2-D 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 620 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

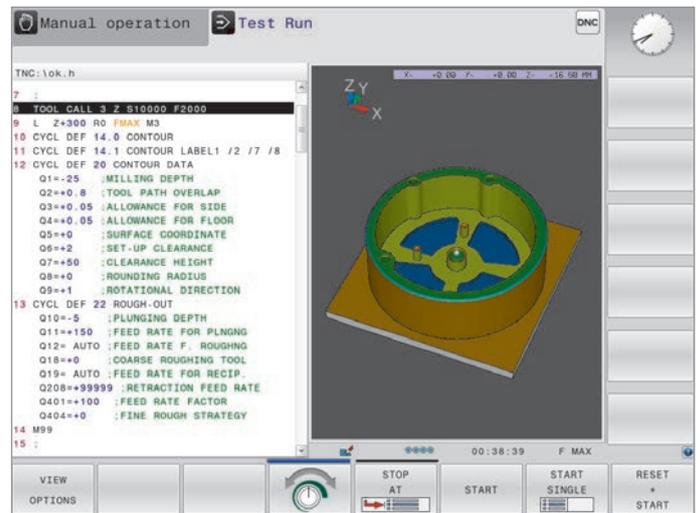
Many machining operations repeat themselves either on the same workpiece or on different workpieces. Once you have programmed a detail, there is no need to program it again. With its subprogram technology, the TNC can save you a great deal of programming time.

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 as often as desired.



Rapid availability of all information

Do you have questions about a programming step but your User's Manual is not at hand? No problem: both the TNC 620 control and the TNC 620 programming station feature TNCguide, a convenient help system that displays the user documentation in a separate window.

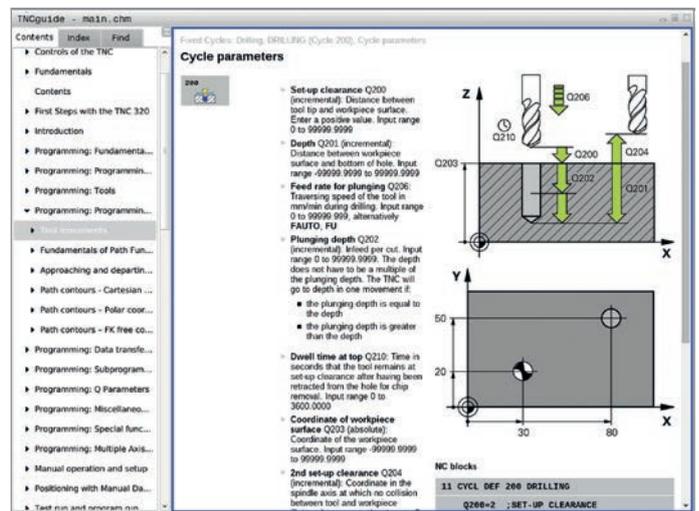
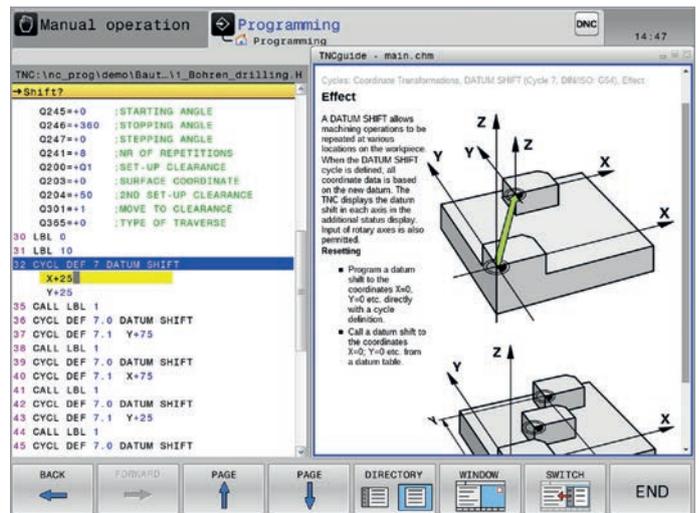
You can activate TNCguide by simply pressing the help key on the TNC keyboard or by clicking any soft key when the mouse pointer has switched to a question mark. This is easily done by clicking on the help icon permanently displayed on the TNC screen.

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 Conversational Programming
- Setup, Testing, and Running NC Programs
- Cycle Programming
- ISO Programming
- TNC 620 Programming Station (installed only with the programming station)



The TNC 620 is designed for exceptional accuracy and surface quality at high machining speeds. A variety of technologies, cycles, and functions operating separately or in concert ensure perfect surface finishes in extremely short machining times:

- Optimized motion control
- Effective jerk reduction
- Dynamic contour look-ahead

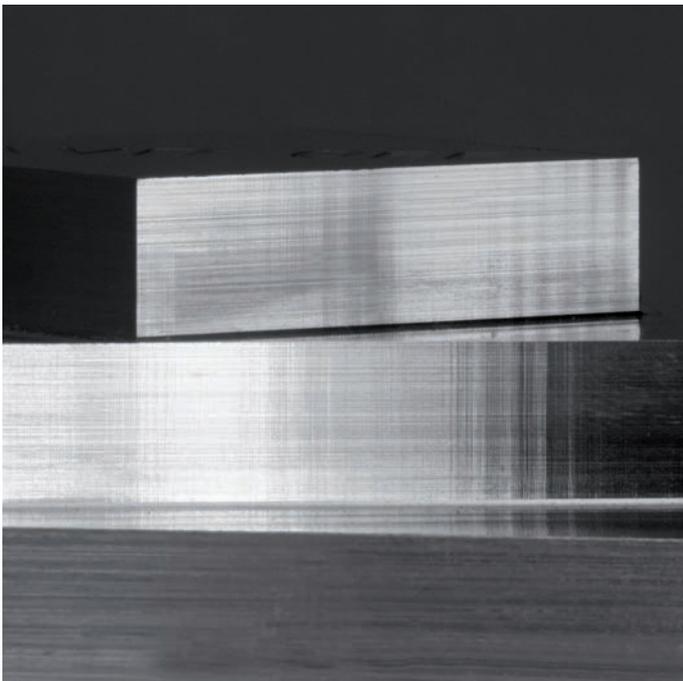
The umbrella term **Dynamic Precision** covers a number of HEIDENHAIN cutting solutions that can dramatically improve the dynamic accuracy of a machine tool. These solutions shed new light on the competing demands of accuracy, high surface quality, and short machining times. The dynamic accuracy of machine tools manifests itself in deviations at the tool center point (TCP). These deviations depend on kinetic quantities such as speed, acceleration, or jerk arising from the vibration of machine components and other factors.

All of these factors contribute to dimensional inaccuracy and the formation of defects on the workpiece surface, thereby impacting both quality and productivity (in the case of quality-related scrap). Dynamic Precision counteracts these problems with intelligent control technology to help improve the quality and dynamic performance of a machine tool. As a result, production time and cost are reduced.

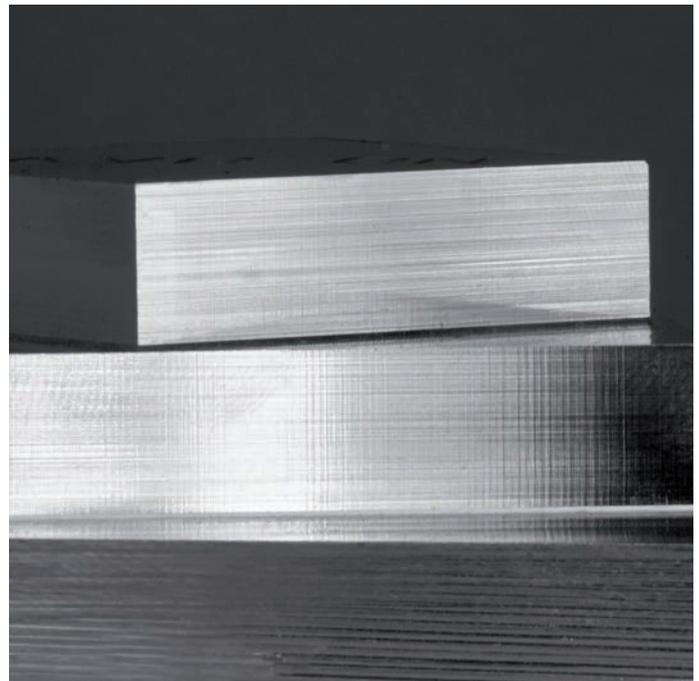


Dynamic Precision contains the following functions, which can be deployed separately or in combination.

Function		Benefits
CTC Cross Talk Compensation	Compensation of acceleration-dependent position errors at the TCP	Greater accuracy during acceleration phases
AVD Active Vibration Damping	Active damping of machine and motor vibrations	A superior surface finish
CTC + AVD	–	Faster and more accurate machining
PAC Position Adaptive Control	Position-dependent adaptation of control parameters	Greater contour fidelity
LAC Load Adaptive Control	Load-dependent adaption of control parameters and the maximum axis acceleration	Higher accuracy whatever the load
MAC Motion Adaptive Control	Motion-dependent adaptation of control parameters	Less vibration and higher maximum acceleration during rapid traverse



Vibration can significantly impair surface quality.



Visibly superior surface quality can be attained with AVD.

Machine any contour slot with trochoidal milling

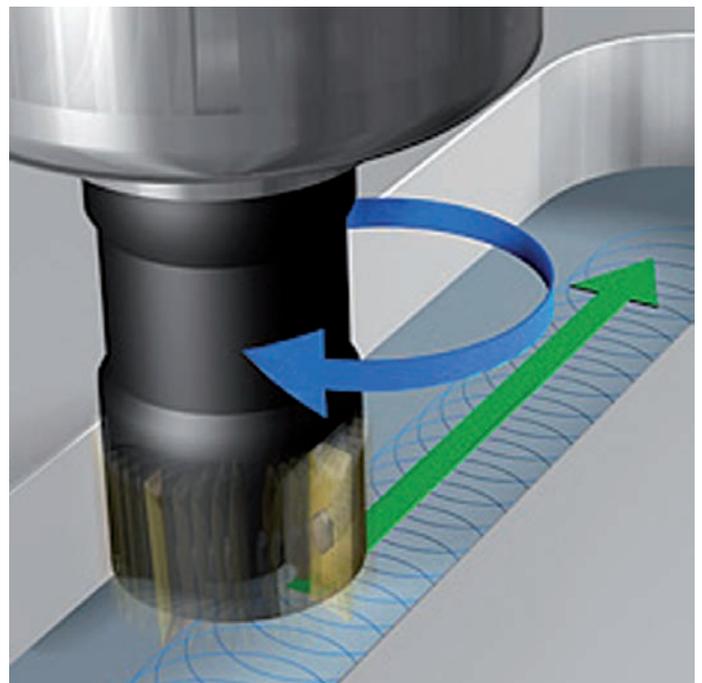
The ability to machine any slot at high efficiency from start to finish is the key benefit of trochoidal milling. In trochoidal milling, roughing is performed with forward linear motion superimposed onto circular strokes. It is often used on high-strength or hardened materials that would otherwise require low infeeds due to the high loads on both tool and machine.

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



Active Chatter Control option (ACC)

Roughing operations (power milling) give rise to strong milling forces. Depending on the rotational speed of the tool, the resonances in the machine tool, and the material removal rate during milling, the tool may begin to "chatter." Chatter puts the machine under heavy strain and causes blemishes on the workpiece surface. Tool wear is also accelerated and less evenly distributed. In extreme cases, the tool may even break.

To reduce chatter susceptibility, HEIDENHAIN now offers a highly effective optional control function called Active Chatter Control (ACC). This option is particularly beneficial during heavy machining:

- Improved cutting performance
- Higher material removal rates (up to 25 % and more)
- Lower forces on the tool, with increased tool life
- Less strain on the machine



Heavy machining without ACC (upper image) and with ACC (lower image)



Optimized roughing processes with OCM (option)

Efficient machining strategies are a key basis for economical NC manufacturing. Roughing processes, in particular, offer great potential for optimization. This is because roughing steps usually take up a sizeable portion of the overall machining time.

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 the Klartext conversational programming language or utilize the convenient CAD Import function. The control then automatically calculates the complex movements required for attaining consistent process conditions.

Advantages of OCM over conventional machining:

- Reduced thermal load on the tool
- Superior chip removal
- Consistent tool contact conditions
 - Higher cutting parameters
 - Higher material removal rates

OCM is an effective, reliable, and convenient way to improve your productivity:

- Perform shopfloor programming for any pocket or island
- Notably higher machining speed
- Considerable reduction in tool wear
- More chips in less time

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

Conventional machining

S5000, F1200, a_p : 5.5 mm

Overlap factor: 5 mm

Machining time: 21 min 35 s

*Tool: VHM end mill \varnothing 10 mm
Workpiece material: 1.4104*

Machining with OCM

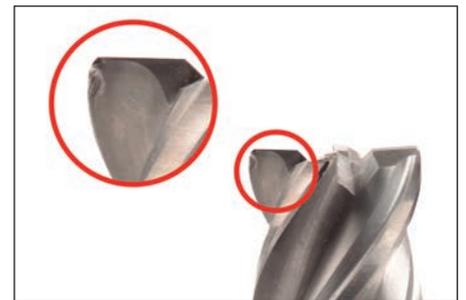
S8000, F4800, a_p : 22 mm

Overlap factor: 1.4 mm

Machining time: 6 min 59 s

*Tool: VHM end mill \varnothing 10 mm
Workpiece material: 1.4104*

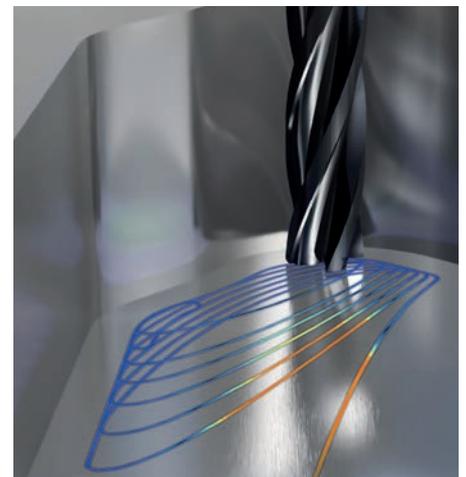
In the following machining example, both the machining time and the tool wear were reduced by a factor of three.



Tool after two parts



Tool after six parts



The TNC 620 programming station

Why a programming station?

It is certainly easy to create a part program at the machine using the TNC 620, even while a different part is being machined. Nevertheless, short reloading times or the level of machine utilization may hinder focused programming work on the shop floor. The TNC 620 programming station lets you program exactly as you would at the machine, but away from all the noise and distractions of the shop.

Creating programs

The programming, testing, and optimizing of HEIDENHAIN Klartext or G-code programs with the programming station substantially reduces machine idle times. This is easy to get used to, since every keystroke feels the same. With the programming station, you program using the same keyboard layout that you use at 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 easily spot contour damage and hidden details.

Training with the programming station

Because the TNC 620 programming station is based on the same software as the TNC 620, it is ideally suited for apprentice and advanced training. Programming is performed on the original keyboard unit, and even the test run feature runs exactly as it would on the machine. This gives the trainee the experience needed to safely work on the real machine.

By being programmable in either Klartext or ISO, the TNC 620 programming station is also well suited for TNC programming training in school settings.

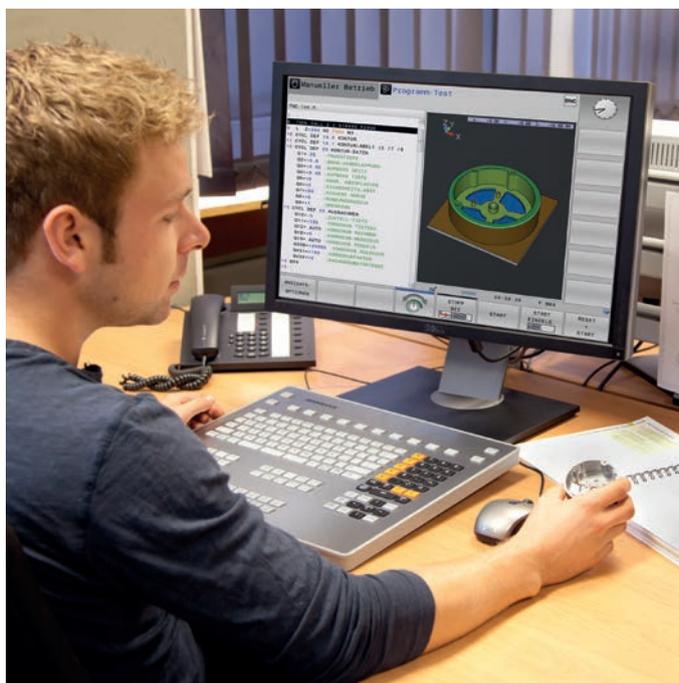
Your workstation

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 620'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. It provides the TNC 620's most important dialog initiation keys.



Programming station with TNC operating panel

Further information:

Comprehensive descriptions of the programming station and a free demo version are available on the Internet at www.heidenhain.de > Products > CNC Controls.

Open to outside information

Processing CAD files with the TNC 620

CAD viewer

With the standardly available CAD viewer software, you can open 3-D CAD models and drawings directly on the TNC 620. 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 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 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 3-D basic rotation of the 3-D model and set a datum with the appropriate 3-D 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 the contour is particularly convenient. You first select any element by clicking it with the mouse. As soon as you have selected a second element, the TNC detects your desired machining direction and initiates automatic contour detection. The TNC automatically selects all clearly identifiable contour elements until the contour closes or branches out. In this manner, you can define extensive contours with just a few mouse clicks. Then you can simply copy the selected contour to an existing Klartext program using the clipboard.



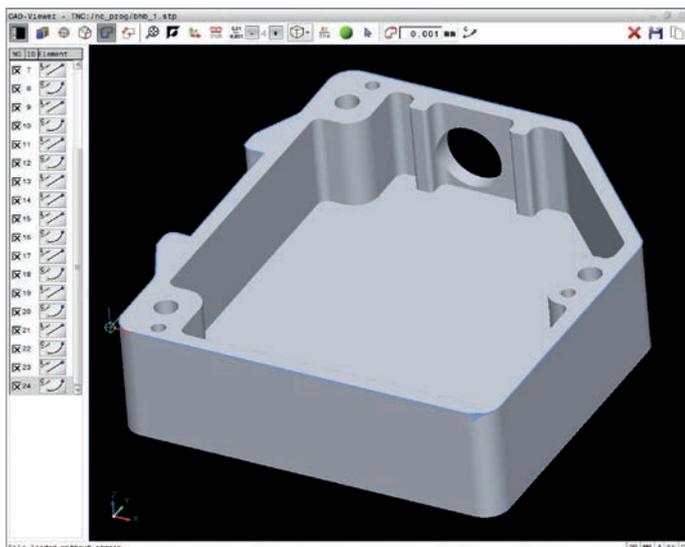
You can also select **machining positions** and save them as point files, especially in order to apply hole positions or starting points for pocket machining. This can be done very easily by selecting the desired area with the mouse. In a pop-up window with a filter function, the TNC displays all of the hole diameters within the area you have selected. To select the desired hole diameters and restrict the number of hole positions, simply 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.

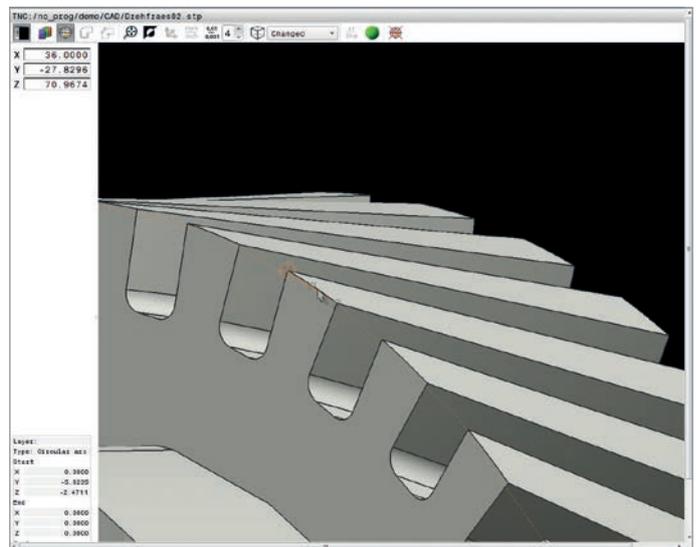
The following positions can be defined as presets:

- 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, regardless of 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.



Contour selection from an imported CAD file



Display of a 3-D model in the CAD viewer

Uniformly digital job management with Connected Machining



Having a smooth exchange of knowledge is crucial for the success of a company. In order for knowledge to be transferred rapidly and without loss, e-mail communication capability is just as important as the continuous availability of electronic production documents or the transfer of data to ERP and production-activity control systems. Tool and raw material inventories, tool data, workholding drawings, CAD data, NC programs and inspection instructions must all be available to machine operators during all shifts. Economical manufacturing thus requires an efficient process chain and a networked control.

With its Connected Machining package of functions, the TNC 620 control integrates itself flexibly into your process chain and helps you optimize the transfer of knowledge within your company. So let your shop

benefit from all of the information available in your company. Connected Machining makes uniformly digital order management possible in networked manufacturing. You thus benefit from the following:

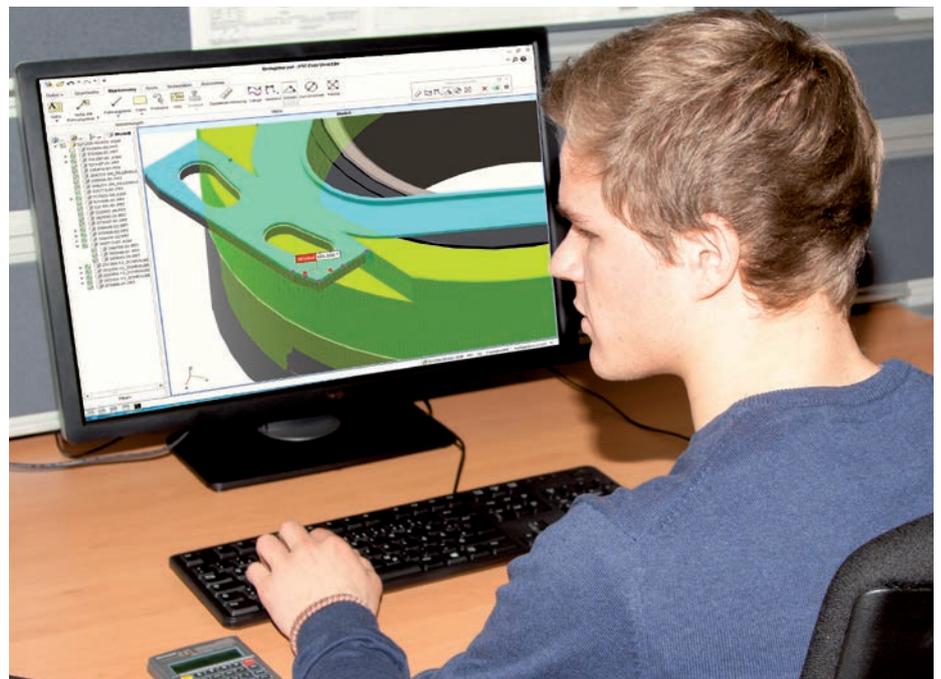
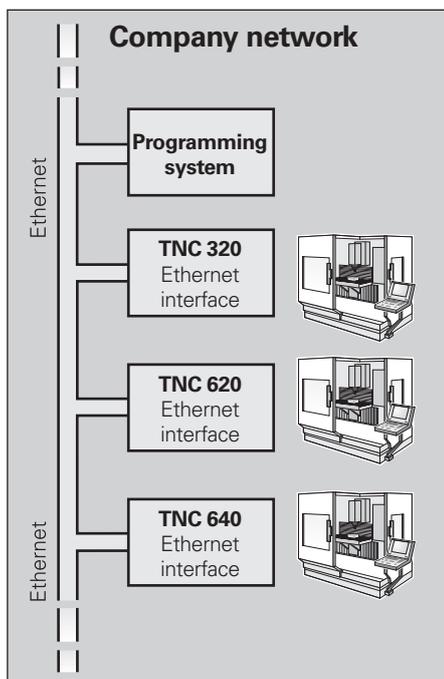
- Ease of data usage
- Time-saving procedures
- Transparent processes

The networked TNC 620

With the functions of Connected Machining, integrate the TNC 620 into your corporate network, and connect the shop floor via the control to PCs, programming stations, and other data storage media in the following areas:

- Design
- Programming
- Simulation
- Production planning
- Production

Even the basic version of the TNC 620 is equipped with a latest-generation Gigabit Ethernet interface in addition to its RS-232-C/V.24 data interface. The TNC 620 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 800 Mbit/s ensures very short transmission times. The TNC 620 thus offers the optimal technological foundation for Connected Machining, the networking of the control in the shop with all production-related areas in your company.



Standard range of functions

In order to be able to use the data that you have transmitted to the control via the standard network connections, the TNC 620 offers you several interesting applications, even in its standard range of functions. The CAD viewer, PDF viewer or Mozilla Firefox web browser enable the simplest form of Connected Machining: access to manufacturing process data right at the control. In this case, operating web-based documentation systems or ERP systems is just as feasible as accessing your e-mail inbox. The following file formats can also be opened directly on the TNC:

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

Data transfer with TNCremo

An enhanced solution for fully digital job management within the context of Connecting Machining is the free-of-charge **TNCremo** PC software. With it, you can do the following (including over the Ethernet interface):

- Transfer remotely stored part programs and tool or pallet tables in both directions
- Start the machine

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

Obtaining operating states

Option 137, State Reporting Interface (SRI), gives you a simple and reliable interface for recording the operating data of your machine. It records the idle times, run times, and fault messages of your machine. SRI also provides historical operating data that can still be retrieved even after the corporate network has been down for hours at a time.

Job-related data

With REMOTE DESKTOP MANAGER (option 133), you can operate a Windows PC right from the TNC 620. From the control, you gain direct access to EDP systems within the process chain and benefit from significantly more efficient setup processes thanks to the reduction in inconvenient legwork between the machine and the office. Technical drawings, CAD data, NC programs, tool data, work instructions, parts lists, and warehouse information are digitally available at the machine. E-mails can be sent and received with ease. Pressing a key on the machine's operating panel conveniently switches you from the screen of the control to the Windows PC interface. The Windows PC can be a computer in the local network or an industrial PC (IPC) in the machine's electrical cabinet.

Component monitoring

During their lifecycle, loaded machine components such as guideways and ball screws exhibit wear, thus causing the quality of axis motion to diminish. This, in turn, affects production quality. With **Component Monitoring** (option 155) and a cycle, the control is able to measure the current machine condition. As a result, any deviations from the machine's shipping condition due to wear and aging can be measured. The machine manufacturer can read and evaluate the data, and react with predictive maintenance, thereby avoiding unplanned machine downtimes.

Detailed data for the optimal organization of production processes

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



StateMonitor: collect and evaluate machine data

StateMonitor collects and visualizes the statuses of production machines. By evaluating important data such as the current machine status, machine messages, override positions, and utilization history, StateMonitor provides in-depth information about the machine's degree of utilization. Based on the collected data, StateMonitor also reveals optimization possibilities. The user can enter comments about machine downtimes and setup times in order to uncover machine-specific and organizational optimization potential. Via the messenger function, StateMonitor notifies the right person by e-mail based on individually combinable machine signals and statuses.

StateMonitor collects and visualizes the following information from the networked machines:

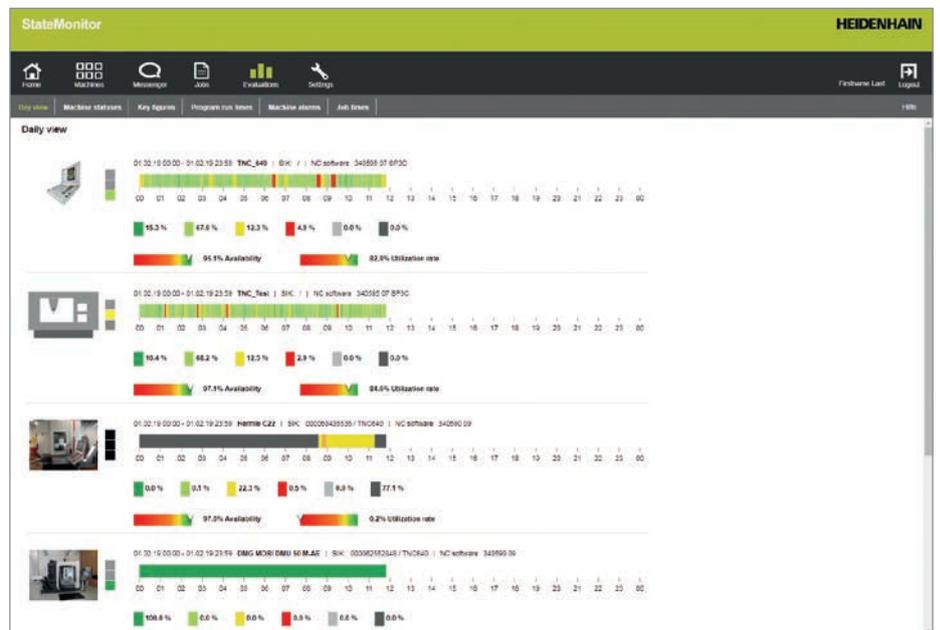
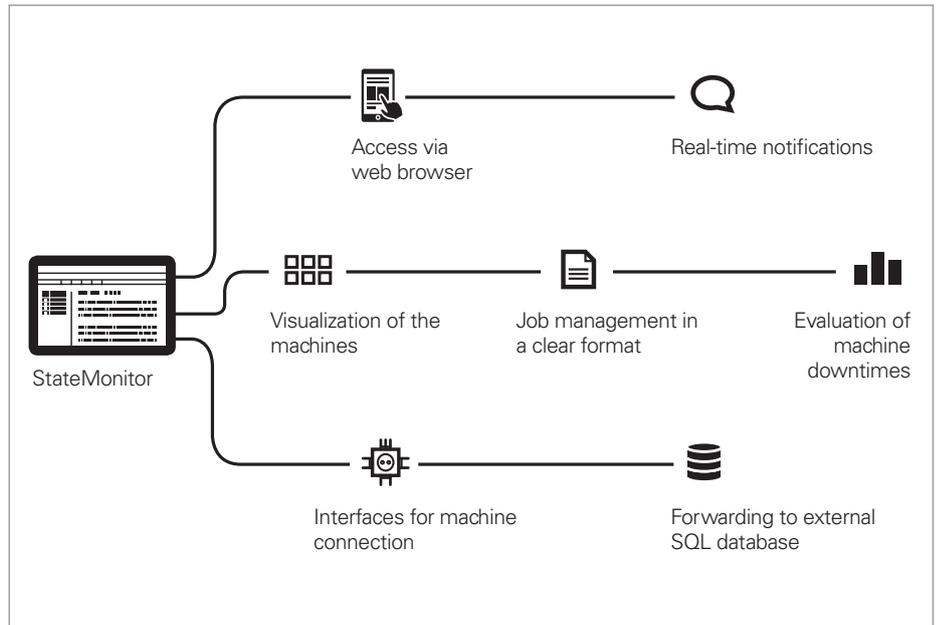
- Operating modes
- Override positions (spindle, rapid traverse, feed rate)
- Program status and program name as well as subprograms, if applicable
- Program run time
- SIK number and software number
- Machine messages

Active support for production planning with comprehensive functions for job data collection:

- Creating and assigning jobs
- Starting and terminating jobs
- Entering setup times and interruptions
- Storing additional job data, such as quantities produced

You can even use StateMonitor to connect machines with various types of controls. The StateMonitor software supports the following protocol types: HEIDENHAIN DNC, OPC UA, MTConnect, and Modbus TCP.

For more information, please contact HEIDENHAIN.



Positioning with an electronic handwheel

Fine motion control of axes

You can set up the workpiece by manually joggling 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 handwheels

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. 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, simply attach it to the machine using its built-in magnets.

The following functions are available on 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 to be defined by the machine tool builder



HR 550

Workpiece measurement

Setup, preset setting, and measuring with touch trigger probes

Workpiece touch probes* from HEIDENHAIN help you keep costs down in the shop and in serial production: when used in conjunction with the TNC 620, 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 either by cable or an infrared beam to the control.

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 and stylus lengths.

* The machine and TNC must be specially adapted by the machine manufacturer.



TS 460

Cable-bound touch probes

For machines requiring manual tool changes, as well as for grinding machines and lathes:

TS 260

- New generation
- Axial or radial cable connection

Wireless touch probes

For machines with an automatic tool changer:

TS 460

- Standard, latest-generation 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 740

- High scanning accuracy
- High reproducibility
- Low probing forces
- Infrared transmission



TS 460 with collision protection

Transceiver unit

Radio or infrared transmission is established between the TS or TT touch probe and the SE transceiver:

SE 660

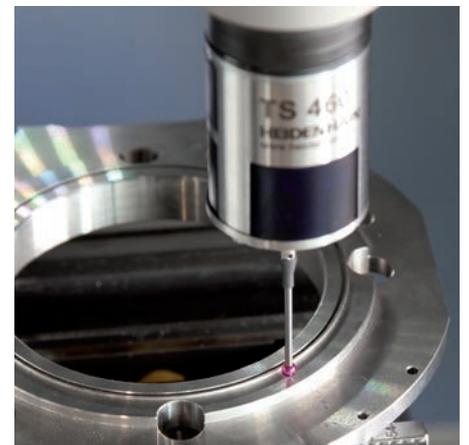
- For radio or infrared transmission (hybrid technology)
- Same SE for TS 460 and TT 460

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



SE 660



Further information:

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

Tool measurement

Measuring length, radius, and wear inside the machine

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 permit tool measurement either before machining or during interruptions.

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

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

To prevent limiting the machining space and 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

TT 460



Further information:

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

Inspecting and optimizing machine accuracy

Easy calibration of rotary axes with KinematicsOpt (option)

Accuracy demands are on the rise, particularly in the realm of 5-axis machining. Complex parts must be manufactured with both precision and reproducible accuracy, including over extended periods of time.

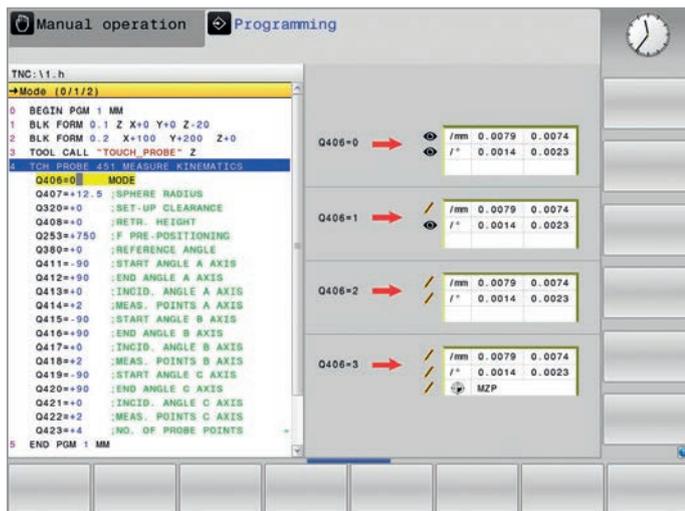
The TNC function **KinematicsOpt** is an important ingredient in helping you meet these high requirements: with a HEIDENHAIN touch probe inserted, a 3-D touch probe cycle measures your machine's rotary axes fully automatically. Measurement is the same regardless of whether the axis is a rotary table, a tilting table, or a swivel head.

To measure the rotary axes, a calibration sphere is fixed at any position on the machine table and probed with the HEIDENHAIN touch probe. Prior to this, you define the resolution of the measurement and the range that you want to measure for each rotary axis.

Based on the measured values, the TNC calculates the static tilting accuracy. The software minimizes the spatial error arising from the tilting movements and, at the end of the measuring process, automatically saves the machine geometry in the respective machine constants of the kinematics description.

Of course, a detailed log file is also available in which the measured and optimized dispersion (measure of the static tilting accuracy) is saved along with the actual measured values and the actual compensation values.

In order to make optimal use of KinematicsOpt, an especially rigid calibration sphere is required. This reduces deformations arising from the probing forces. For this reason, HEIDENHAIN offers calibration spheres with highly rigid holders available in various lengths.



Overview

Specifications

Specifications	Standard	Option	
Components	✓		MC main computer with operating panel and integrated 15-inch or 19-inch screen (portrait orientation) for multitouch operation MC main computer with separate TE 730 or TE 735 operating panel and integrated 15-inch screen
Operating system	✓		HEROS 5 real-time operating system for machine control
Memory	✓		1.8 GB (on CFR compact flash memory card) for NC programs
Input resolution and display step	✓ ✓		Linear axes: down to 0.01 µm Rotary axes: down to 0.00001°
Input range	✓		Maximum 999 999 999 mm or 999 999 999°
Interpolation	✓ ✓ ✓	9 8	Linear in 4 axes Linear in up to 5 axes (export license required) Circular in 2 axes Circular in 3 axes with tilted working plane Helical: superimposition of circular and straight paths
Block processing time	✓		1.5 ms (3-D straight line without radius compensation)
Axis feedback control	✓ ✓		Position loop resolution: signal period of the position encoder/1024 Cycle time of interpolator: 3 ms
Spindle speed	✓		Maximum 100000 rpm (with 2 pole pairs)
Error compensation	✓ ✓		Linear and nonlinear axis error, backlash, reversal peaks during circular movements, thermal expansion Static friction, sliding friction, reversal error
Data interfaces	✓ ✓ ✓ ✓	18	RS-232-C/V.24 max. 115 kbit/s Extended data interface with LSV-2 protocol for remote operation of the TNC 620 over the data interface with the HEIDENHAIN software TNCremo or TNCremoPlus Gigabit Ethernet interface 1000BASE-T 5 x USB (1 x front USB 2.0; 4 x back panel USB 3.0) HEIDENHAIN-DNC for communication between a Windows application and TNC (DCOM interface)
Diagnostics	✓		Fast and simple troubleshooting through integrated diagnostic aids
Ambient temperature	✓ ✓		Operation: +5 °C to +40 °C Storage: -20 °C to +60 °C

Overview

User functions

User functions	Standard	Option	
Short description	✓ ✓	0/1	Basic version: 3 axes plus closed-loop spindle 1st or 2nd additional axis for 4 or 5 axes plus spindle Digital current and speed control
Program entry	✓ ✓	42	In HEIDENHAIN Klartext conversational programming language ISO (<i>with integrated operating panel</i> : via soft keys or external standard USB keyboard; <i>with separate operating panel</i> : via ASCII keyboard) Importing of contours or machining positions from CAD files (STP, IGS, DXF) and saving them as a Klartext contour program or Klartext point tables
Position entry	✓ ✓ ✓		Nominal positions for straight lines and arcs in Cartesian coordinates or polar coordinates Incremental or absolute dimensions Display and entry in mm or inches
Tool compensation	✓	21 9	Tool radius in the working plane and tool length Radius-compensated contour look-ahead for up to 99 blocks (M120) Three-dimensional tool-radius compensation for later modification of tool data without the need for program recalculation
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 point Relative to the tool's cutting edge
Parallel operation	✓		Program creation with graphical support while another program is running
3-D machining	✓	9 9 9 9	Motion control with highly smoothed jerk 3-D tool compensation through surface normal vectors Keeping the tool perpendicular to the contour Tool radius compensation perpendicular to the tool direction Manual traverse in the active tool-axis system
Rotary table machining		8 8	Programming of contours for machining on a cylinder 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
Approaching and departing the contour	✓ ✓		Via straight line: tangential or perpendicular Via circular arc
FK free contour programming		19	FK free contour programming in the HEIDENHAIN Klartext conversational programming language with graphical support for workpiece drawings not dimensioned for NC
Program jumps	✓ ✓ ✓		Subprograms Program-section repeats Any program as a subprogram
Teach-in	✓		Application of actual positions directly in the NC program

User functions	Standard	Option	
Fixed cycles	✓	19 19 19 19 19 19 19 19 19 19 19 167	Drilling, tapping (with or without floating tap holder), rectangular and circular pockets Peck drilling, reaming, boring, counterboring, centering Milling of internal and external threads Clearing of level and oblique surfaces Complete machining of rectangular and circular pockets, rectangular and circular studs Complete machining of straight and circular slots Cartesian and polar point patterns and point patterns for DataMatrix code Contour train, contour pocket Contour slot with trochoidal milling Engraving cycle: engraving of text or numbers in a straight line or arc Integratability of OEM cycles (special cycles developed by the machine manufacturer) Optimized Contour Milling (OCM) cycles: optimization of roughing processes
Coordinate transformations	✓	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 Functions for text processing
Programming aids	✓ ✓ ✓ ✓ ✓		Calculator Complete list of all current error messages Context-sensitive help function for error messages TNCguide: the integrated help system; user information available directly on the TNC 620 Graphical support for the programming of cycles Comment and structure blocks in the NC program
Program verification graphics Display modes		20 20 20	Graphical simulation of the machining operation, even while another program is being executed 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		20 20	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 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 saving any presets
Datum tables	✓		Multiple datum tables for storing workpiece-specific datums
Pallet tables		22 154	Workpiece-oriented execution of pallet tables with any number of entries for selection of pallets, NC programs, and datums Planning the production process with Batch Process Manager

User functions (continued)

Accessories

User functions	Standard	Option	
Touch probe cycles*		17	Touch probe calibration
		17	Manual or automatic compensation of workpiece misalignment
		17	Manual or automatic preset setting
		17	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 standardized CAD file formats on the TNC

* Automatically enabled for HEIDENHAIN touch probes.

Accessory	
Electronic handwheels	<ul style="list-style-type: none"> • One HR 510 FS portable handwheel, or • One HR 520 FS portable handwheel with display, or • One HR 550 FS portable wireless handwheel with display, or • One HR 130 panel-mounted handwheel
Workpiece measurement	<ul style="list-style-type: none"> • TS 260: workpiece touch probe with cable connection, or • TS 248: workpiece touch probe with reduced deflection force, or • TS 460: triggering workpiece touch probe with radio or infrared transmission, or • TS 642: workpiece touch probe with infrared transmission, or • TS 740 triggering workpiece touch probe with infrared transmission
Tool measurement	<ul style="list-style-type: none"> • TT 160: triggering tool touch probe, or • TT 460: triggering tool touch probe with radio or infrared transmission
Programming station	Control software for PCs for programming, archiving, and training <ul style="list-style-type: none"> • 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 with virtual keyboard or PC keyboard—free of charge
Software for PCs	<ul style="list-style-type: none"> • 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 • StateMonitor: software for recording, evaluating, and visualizing machine data

Options

Option number	Option	As of NC software 81776x-	ID	Comment
0 1 2 3	Additional Axis	01	354540-01 353904-01 353905-01 367867-01	Additional control loop 1 Additional control loop 2 Additional control loop 3 Additional control loop 4
8	Advanced Function Set 1	01	617920-01	Rotary table machining <ul style="list-style-type: none"> • Programming of contours for machining on a cylinder as if in two axes • Feed rate in mm/min Interpolation: circular in three axes with tilted working plane Coordinate transformation: tilting the working plane, PLANE function
9	Advanced Function Set 2	01	617921-01	Interpolation: straight line in more than four axes (export license required) 3-D machining <ul style="list-style-type: none"> • 3-D tool compensation via surface normal vectors • Alteration of the swivel head angle via the electronic handwheel during program run without changing the position of the tool center point (TCPM = Tool Center Point Management); only with option 21 • Keeping the tool perpendicular to the contour • Tool radius compensation perpendicular to the tool direction • Manual traverse in the active tool-axis system
17	Touch Probe Functions	01	634063-01	Touch probe cycles <ul style="list-style-type: none"> • Compensation of workpiece misalignment, setting of presets • Automatic tool and workpiece measurement • Touch probe input enabled for non-HEIDENHAIN touch probes
18	HEIDENHAIN DNC	01	526451-01	Communication with external PC applications over COM component
19	Advanced Programming Features	01	628252-01	FK free contour programming Fixed cycles <ul style="list-style-type: none"> • Peck drilling, reaming, boring, counterboring, centering • Milling of internal and external threads • Clearing of level and oblique surfaces • Multi-operation machining of straight and circular slots • Multi-operation machining of rectangular and circular pockets • Cartesian and polar point patterns and point patterns for DataMatrix code • Contour train, contour pocket—also with contour-parallel machining • Contour slot with trochoidal milling • Special cycles developed by the machine tool builder can be integrated
20	Advanced Graphic Features	01	628253-01	Program-verification graphics, program-run graphics Plan view, projection in three planes, 3-D view, 3-D line graphic
21	Advanced Function Set 3	01	628254-01	Tool compensation <ul style="list-style-type: none"> • Radius-compensated contour look-ahead for up to 99 blocks (LOOK AHEAD) 3-D machining <ul style="list-style-type: none"> • Superimposing handwheel positioning during program run
22	Pallet Management	01	628255-01	Pallet management
24	Gantry Axes	01	634621-01	Gantry axes via master-slave torque control

Options (continued)

Option number	Option	As of NC software 81776x-	ID	Comment
42	CAD Import	05	526450-01	Importing of contours from 3-D models
46	Python OEM Process	01	579650-01	Python application on the TNC
48	KinematicsOpt	01	630916-01	Touch-probe cycles for the automatic measurement of rotary axes
49	Double Speed	01	632223-01	Short control-loop cycle times for direct drives
93	Extended Tool Management	02	676938-01	Extended tool management for more information
133	Remote Desk. Manager	01	894423-01	Display and remote operation of external computer units (e.g., a Windows PC)
137	State Reporting Interface	06	1232242-01	Obtaining operating states
141	Cross Talk Comp.	01	800542-01	CTC: compensation of axes coupling
142	Pos. Adapt. Control	01	800544-01	PAC: position-dependent adaptation of the control parameters
143	Load Adapt. Control	01	800545-01	LAC: load-dependent adaptation of the control parameters
144	Motion Adapt. Control	01	800546-01	MAC: motion-dependent adaptation of control parameters
145	Active Chatter Control	01	800547-01	ACC: active suppression of chatter during heavy machining
146	Active Vibration Damping	01	800548-01	AVD: active suppression of vibration
154	Batch Process Manager	05	121952-01	Depiction of pallet management in a clear format
155	Component Monitoring	06	1226833-01	Function for the monitoring of components for overload and wear
160	Integrated FS: Basic	07	1249928-01	Gen 3 exclusive: <ul style="list-style-type: none"> • Enables 4 safe control loops • Enables functional safety (FS)
161	Integrated FS: Full	07	1249929-01	Gen 3 exclusive: <ul style="list-style-type: none"> • Enables the maximum number of safe control loops (≥ 10) • Enables functional safety (FS)
162	Add. FS Ctrl. Loop 1	07	1249930-01	Gen 3 exclusive: Additional FS control loop 1
163	Add. FS Ctrl. Loop 2	07	1249931-01	Additional FS control loop 2
164	Add. FS Ctrl. Loop 3	07	1249932-01	Additional FS control loop 3
165	Add. FS Ctrl. Loop 4	07	1249933-01	Additional FS control loop 4
166	Add. FS Ctrl. Loop 5	07	1249934-01	Additional FS control loop 5
167	Optimized Contour Milling (OCM)	10	1289547-01	OCM: Optimized Contour Milling

Comparison of controls

Comparison of controls	TNC 620 NC SW 81760x07	TNC 640 NC SW 34059x10
Area of application	Standard milling	High-end milling / turning / grinding
• Simple machining centers (up to 8 control loops, max. 2 of which are spindles)	✓	✓
• Machine tools / machining centers (up to 24 control loops, max. 4 of which are spindles)	–	✓
Program entry		
• HEIDENHAIN Klartext conversational programming language	✓	✓
• ISO programming	✓	✓
• CAD Import	Option	Option
• CAD viewer	✓	✓
• FK free contour programming	Option	✓
• Extended milling and drilling cycles	Option	✓
• OCM cycles	Option	Option
• Turning cycles	–	Option
• Functions for grinding	–	Option
NC program memory	1.8 GB	> 21 GB
5-axis and high-speed machining	Option	Option
Block processing time	1.5 ms	0.5 ms
Input resolution and display step (standard)	0.01 µm	0.01 µm
New design of the screen and keyboard	15-inch screen/ 19-inch touchscreen (portrait)	19-inch / 24-inch touchscreen
Touchscreen	✓	✓
Adaptive Feed Control (AFC)	–	Option
Active Chatter Control (ACC)	Option	Option
Dynamic Collision Monitoring (DCM)	–	Option
KinematicsOpt	Option	Option
KinematicsComp	–	Option
Touch probe cycles	Option	✓
Pallet Management	Option	✓
Parallel axis function	✓	✓

✓ Standard
– Not available

HEIDENHAIN

DR. JOHANNES HEIDENHAIN GmbH

Dr.-Johannes-Heidenhain-Straße 5

83301 Traunreut, Germany

☎ +49 8669 31-0

FAXI +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 FAXI 08669 32-3132 E-Mail: hd@heidenhain.de	ES	FARRESA ELECTRONICA S.A. 08028 Barcelona, Spain www.farresa.es	PL	APS 02-384 Warszawa, Poland www.heidenhain.pl
	HEIDENHAIN Technisches Büro Nord 12681 Berlin, Deutschland ☎ 030 54705-240	FI	HEIDENHAIN Scandinavia AB 01740 Vantaa, Finland www.heidenhain.fi	PT	FARRESA ELECTRÓNICA, LDA. 4470 - 177 Maia, Portugal www.farresa.pt
	HEIDENHAIN Technisches Büro Mitte 07751 Jena, Deutschland ☎ 03641 4728-250	FR	HEIDENHAIN FRANCE sarl 92310 Sèvres, France www.heidenhain.fr	RO	HEIDENHAIN Reprezentantă Romania Braşov, 500407, Romania www.heidenhain.ro
	HEIDENHAIN Technisches Büro West 44379 Dortmund, Deutschland ☎ 0231 618083-0	GB	HEIDENHAIN (G.B.) Limited Burgess Hill RH15 9RD, United Kingdom www.heidenhain.co.uk	RS	Serbia → BG
	HEIDENHAIN Technisches Büro Südwest 70771 Leinfelden-Echterdingen, Deutschland ☎ 0711 993395-0	GR	MB Milionis Vassilis 17341 Athens, Greece www.heidenhain.gr	RU	OOO HEIDENHAIN 115172 Moscow, Russia www.heidenhain.ru
	HEIDENHAIN Technisches Büro Südost 83301 Traunreut, Deutschland ☎ 08669 31-1337	HR	Croatia → SL	SE	HEIDENHAIN Scandinavia AB 12739 Skärholmen, Sweden www.heidenhain.se
AR	NAKASE SRL. B1653AOX Villa Ballester, Argentina www.heidenhain.com.ar	HU	HEIDENHAIN Kereskedelmi Képviselet 1239 Budapest, Hungary www.heidenhain.hu	SG	HEIDENHAIN PACIFIC PTE LTD Singapore 408593 www.heidenhain.com.sg
AT	HEIDENHAIN Techn. Büro Österreich 83301 Traunreut, Germany www.heidenhain.de	ID	PT Servitama Era Toolsindo Jakarta 13930, Indonesia E-mail: ptset@group.gts.co.id	SK	KOPRETINA TN s.r.o. 91101 Trenčin, Slovakia www.kopretina.sk
AU	FCR MOTION TECHNOLOGY PTY LTD Laverton North Victoria 3026, Australia E-mail: sales@fcrmotion.com	IL	NEUMO VARGUS MARKETING LTD. Holon, 58859, Israel E-mail: neumo@neumo-vargus.co.il	SL	NAVO d.o.o. 2000 Maribor, Slovenia www.heidenhain.si
BE	HEIDENHAIN NV 1760 Roosdaal, Belgium www.heidenhain.be	IN	HEIDENHAIN Optics & Electronics India Private Limited Chetpet, Chennai 600 031, India www.heidenhain.in	TH	HEIDENHAIN (THAILAND) LTD Bangkok 10250, Thailand www.heidenhain.co.th
BG	ESD Bulgaria Ltd. Sofia 1172, Bulgaria www.esd.bg	IT	HEIDENHAIN ITALIANA S.r.l. 20128 Milano, Italy www.heidenhain.it	TR	T&M Mühendislik San. ve Tic. LTD. ŞTİ. 34775 Y. Dudullu – Umraniye-Istanbul, Turkey www.heidenhain.com.tr
BR	HEIDENHAIN Brasil Ltda. 04763-070 – São Paulo – SP, Brazil www.heidenhain.com.br	JP	HEIDENHAIN K.K. Tokyo 102-0083, Japan www.heidenhain.co.jp	TW	HEIDENHAIN CO., LTD. Taichung 40768, Taiwan www.heidenhain.com.tw
BY	GERTNER Service GmbH 220026 Minsk, Belarus www.heidenhain.by	KR	HEIDENHAIN Korea Ltd. Anyang-si, Gyeonggi-do, 14087 South Korea www.heidenhain.co.kr	UA	Gertner Service GmbH Büro Kiev 02094 Kiev, Ukraine www.heidenhain.ua
CA	HEIDENHAIN CORPORATION Mississauga, Ontario L5T2N2, Canada www.heidenhain.com	MX	HEIDENHAIN CORPORATION MEXICO 20290 Aguascalientes, AGS., Mexico E-mail: info@heidenhain.com	US	HEIDENHAIN CORPORATION Schaumburg, IL 60173-5337, USA www.heidenhain.com
CH	HEIDENHAIN (SCHWEIZ) AG 8603 Schwerzenbach, Switzerland www.heidenhain.ch	MY	ISOSERVE SDN. BHD. 43200 Balakong, Selangor E-mail: sales@isoserve.com.my	VN	AMS Co. Ltd HCM City, Vietnam E-mail: davidgoh@amsvn.com
CN	DR. JOHANNES HEIDENHAIN (CHINA) Co., Ltd. Beijing 101312, China www.heidenhain.com.cn	NL	HEIDENHAIN NEDERLAND B.V. 6716 BM Ede, Netherlands www.heidenhain.nl	ZA	MAFEMA SALES SERVICES C.C. Kyalami 1684, South Africa www.heidenhain.co.za
CZ	HEIDENHAIN s.r.o. 102 00 Praha 10, Czech Republic www.heidenhain.cz	NO	HEIDENHAIN Scandinavia AB 7300 Orkanger, Norway www.heidenhain.no		
DK	TPTEKNIK A/S 2670 Greve, Denmark www.tp-gruppen.dk	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		