



HEIDENHAIN

araxe

72, rue Yves le Coz
78000 VERSAILLES

tél : 01 30 21 48 49

fax : 01 39 51 16 33

<http://www.araxe.com>

contact@araxe.com

HEIDENHAIN



TNC 128

The Compact Straight-Cut Control for Milling, Drilling and Boring Machines



This brochure describes the functions and specifications of the TNC 128 with NC software 771841-05.

Contents

The TNC 128...

Where can it be used?	Compact and convenient – The control for simple milling, drilling and boring machines	4
What does it look like?	Well designed and user friendly – The TNC 128 in dialog with the user	6
What can it do?	Minimize setup and non-machining time – The TNC 128 makes setup easy	8
How is it programmed?	Programming, editing, testing – The TNC 128 opens endless possibilities – Graphic support in any situation	10
	Programming in the workshop – Field-proven cycles for recurring operations – Reusing programmed contour elements – Fast availability of all information	12
	Open for communication – Fast data transfer with the TNC 128	16
Are there any accessories?	Positioning with the electronic handwheel – Delicate axis traverse	17
	Workpiece measurement – Setup, presetting and measuring with touch trigger probes	18
	Tool measurement – Measuring length, radius and wear directly in the machine	19
... At a glance	Overview – User functions – Options – Accessories – Specifications	20

Compact and convenient

–The control for simple milling, drilling and boring machines

Controls from HEIDENHAIN have been proving their reliability on various types of machines for over 35 years. In the early days of CNC machining, most operations were performed with simple CNC milling machines that were capable of cutting with no more than three servo-controlled axes at only moderate traversing speeds. Nowadays complex machining centers are in operation, CNC machines are linked together, and machine tools are equipped with automated loading systems. Less complex operations continue to be performed on simple CNC milling machines. The TNC 128 straight cut control was conceived for exactly this purpose. Its scope of functions is perfectly tailored to simple CNC machines, and it is suitable for machines with central drive as well as those with independent drive.

Workshop-oriented programming

Workshop-oriented programming is an important attribute of the TNC 128, particularly since the TNC controls are rooted in the workshop.

In the past years, machining processes have become more complex, and the machines themselves more powerful. Throughout, the basic operation of the TNC controls has remained the same despite their continued development and improvement.

The proven operation is also the basis for the user-friendly programming of the TNC 128, since it assists you during program creation with help graphics, practical prompts, machining cycles and cycles for coordinate transformation.

Easy to operate

For simple work, such as face milling, you need not write a program on the TNC 128. It is just as easy to operate the machine manually by pressing the axis keys or—for maximum sensitivity—using the electronic handwheel.

Offline program creation

The TNC 128 can be programmed remotely just as well. Its Gigabit Ethernet interface guarantees very short transfer times, even of long programs.



The TNC 128 is compact and easy to read

The TNC 128 is a compact but versatile straight-cut control for three servo axes and servo spindle. A further servo axis is an option. Thanks to its simple 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
- Machine building
- Prototypes and pilot plants
- Repair departments
- Training and education facilities

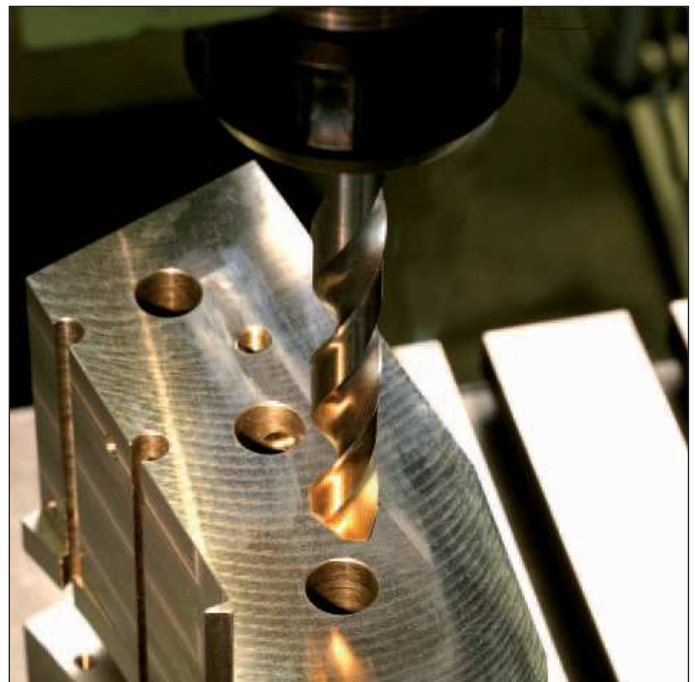
It also offers the applicable features both necessary and helpful for:

Universal milling machines

- Milling cycles for rectangular pockets, rectangular studs and for face milling
- Fast presetting with HEIDENHAIN touch probes

Drilling and boring machines

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



Well designed and user friendly

– The TNC 128 in dialog with the user

The screen

The 12.1-inch TFT color flat-panel display shows a clear overview of all relevant information for programming, operating and inspecting the machine tool and control, such as program blocks, comments and error messages. More information is provided through graphic support during program entry, test run and actual machining.

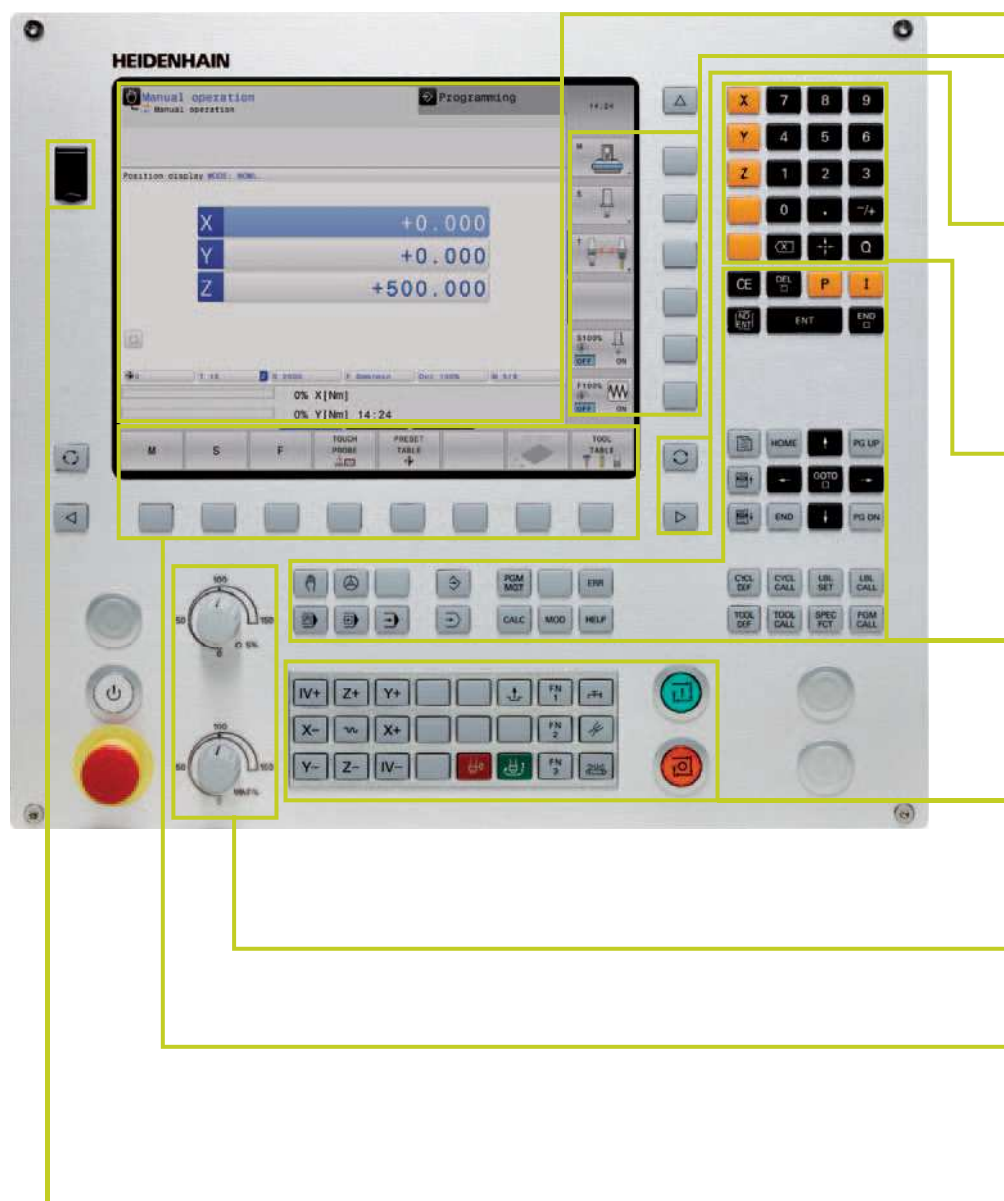
The selectable “split screen” display shows the part program blocks in one half of the screen and the graphics or the status display in the other half.

During the course of the program, status displays will always offer information on tool position, the current program, active cycles and coordinate transformations, and other data. The TNC 128 even shows the current machining time.

The keyboard

As with all TNCs from HEIDENHAIN, the operating panel is oriented to the programming process. The well thought-out configuration of keys facilitates program input. Simple words and abbreviations or unambiguous symbols clearly indicate each key's function. Certain functions of the TNC 128 are available by soft key.

The integrated **machine operating panel** features easily exchangeable snap-on keys that allow simple adaptation to the respective machine configuration. You use the **override potentiometers** to make delicate adjustments of feed-rate, rapid traverse and spindle speed.



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

PLC function keys (soft keys) for machine functions

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

Axis-selection keys and numeric keypad

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

Machine operating panel with snap-on keys

Override potentiometers for feed rate, rapid traverse and spindle speed

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

USB port for additional data storage or pointing devices

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

Durable

The high-quality stainless steel design of the TNC 128 features a special protective coating and is therefore highly resistant to soiling and wear.

Smooth

The rectangular, slightly rounded keys are pleasant to the touch and reliable in operation. Their inscriptions do not wear off, even under extreme workshop conditions.

Versatile

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

Sensitive

With the handy control knobs you can individually adjust the feed-rate and spindle speed.

Communicative

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

Flexible

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

Reliable

The elevated key bed of the machine operating panel prevents accidental actuation.



Minimize setup and non-machining time

– The TNC 128 makes setup easy

Before you can begin machining, you must first clamp the tool and set up the machine, find the position of the workpiece on the machine, and set the workpiece reference point. Without support from the control this is often a time-consuming procedure, but it is indispensable. After all, any error directly reduces the machining accuracy. Particularly in small and medium-sized production runs, as well as for very large workpieces, setup times become quite a significant factor.

Here the TNC 128 shows its strengths: With its practice-oriented setup features it supports the operator and helps to reduce non-machining time. Together with the **touch probes**, the TNC 128 offers various probing functions for presetting as well as measurement of the workpiece and the tool.

Delicate manual traverse

For setup, you can use the direction keys to move the machine axes manually or in incremental jog. A simpler and more reliable way, however, is to use the electronic handwheels from HEIDENHAIN (see page 17). With the handwheels you are always close to the action, enjoy a close-up view of the setup process, and can control the infeed responsively and precisely.

Adapting the probing velocity

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

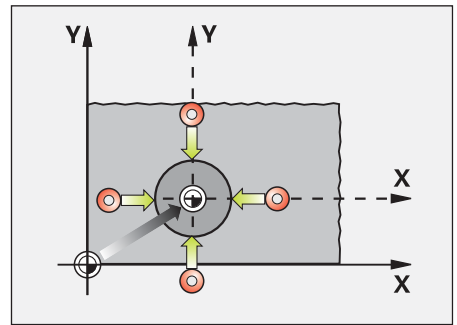
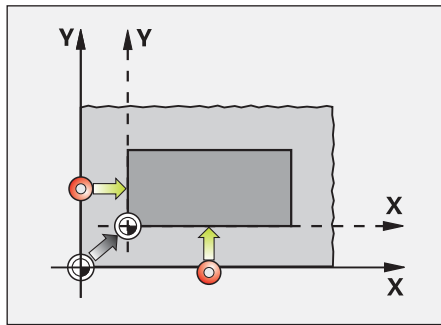


Setting datums

You can use a reference point to assign a defined value in the TNC display to any workpiece position. Finding this point quickly and reliably reduces nonproductive time and increases machining accuracy.

The TNC 128 features probing functions for presetting. Once found, you can save these datums

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



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

Datum management with the preset table

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

Any number of reference points can be saved in the preset management. To permanently save fixed datums in the machine working space, you can also provide individual lines with write protection. There are two possibilities for rapid saving of the reference points:

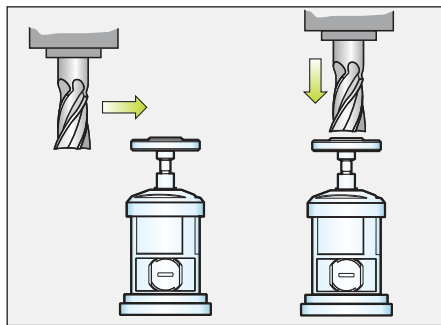
- In the Manual mode by soft key
- By using the probing functions

Saving datums

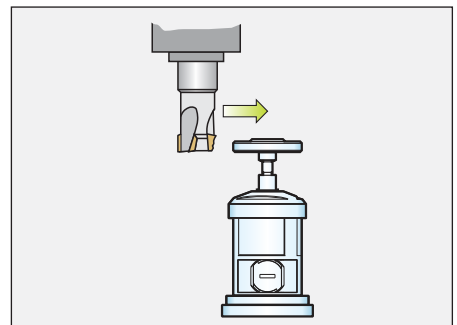
In datum tables, you can save positions or values given measured with respect to the workpiece. Datums are always relative to the active reference point.

Tool measurement and automatic compensation of tool data

Together with the TT 160 for tool measurement (see page 19), the TNC 128 makes it possible to measure tools while they are in the machine spindle. The TNC 128 saves the ascertained values of tool length and radius in the central tool file. By inspecting the tool you can quickly and directly measure wear or breakage to prevent scrap or rework.



Tool length and radius measurement



Measuring tool wear

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

DOC: Text width 16 TNC:\table\preset.pr

0% X [Nm]
0% Y [Nm] 09:16

X: +0.000
Y: +0.000
Z: +500.000

Mode: NOML. T 15 S 2000
F 0mm/min Ovr 100% M 5/9

BEGIN END PAGE PAGE CHANGE ACTIVATE END
↑ ↓ ↑ ↓ PRESET PRESET

Programming, editing, testing

– The TNC 128 opens endless possibilities

The TNC 128 is just as universal in application as it is flexible in machining and programming.

Positioning with Manual Data Input

You can start working with the TNC 128 even before writing a complete part program. Simply machine a part step by step—switching as you want between manual operation and automatic positioning.

Programming at the machine

HEIDENHAIN controls are workshop oriented, which means that they were conceived for programming right at the machine. With **Klartext conversational programming** you can forget about memorizing G codes. Instead you program using dedicated keys and soft keys, which precisely indicate the respectively associated function. You initiate a HEIDENHAIN Klartext dialog with a keystroke and the TNC immediately begins to support you actively in your work. Unambiguous questions and prompts help you enter all the required information.

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

Managing programs offline

The TNC 128 is also well equipped for external access. Through its interfaces it can be integrated into networks and connected with programming stations or other data storage devices.



– Graphic support in any situation

Programming graphics

The two-dimensional programming graphics give you additional security: while you are programming, the TNC 128 draws every entered traverse command on the screen. You can select among the plan view, side view and front view.

Test graphics

To play it safe before running a program, the TNC 128 can graphically simulate the machining of the workpiece. The TNC 128 can display the simulation in the following ways:

- In a plan view with different shades of depth
- In three planes (as in the workpiece drawing)
- In a solid model, 3-D view

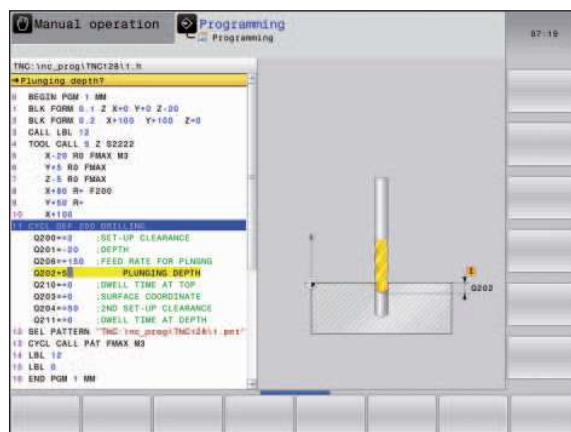
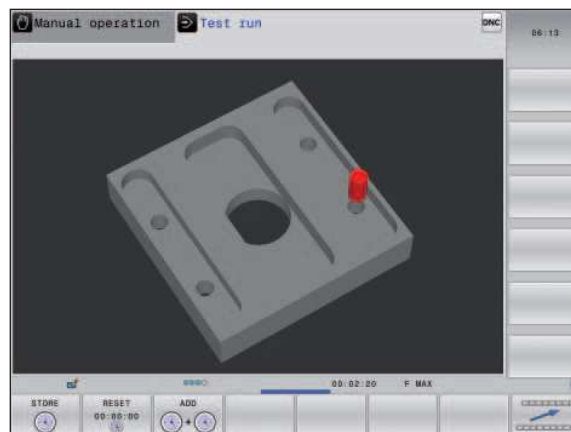
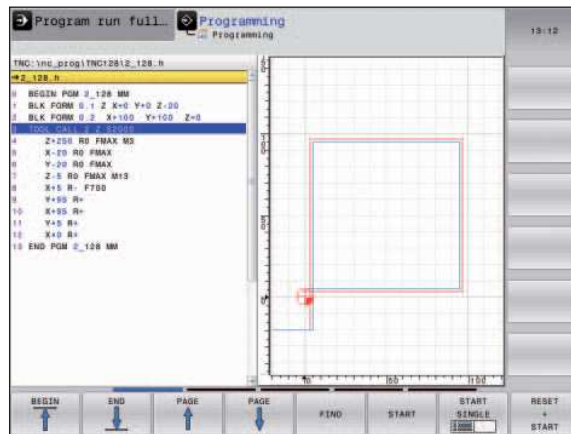
Details can be displayed in magnification. In addition, the TNC 128 indicates the calculated machining time in hours, minutes and seconds. In the 3-D view, the TNC features a measuring function. You can position the mouse pointer anywhere in the graphic to see the coordinates.

Program-run graphics

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

Help graphics

During cycle programming in plain language, the TNC shows a separate illustration for each parameter. This makes it easier to understand the function and accelerates programming.



Programming in the workshop

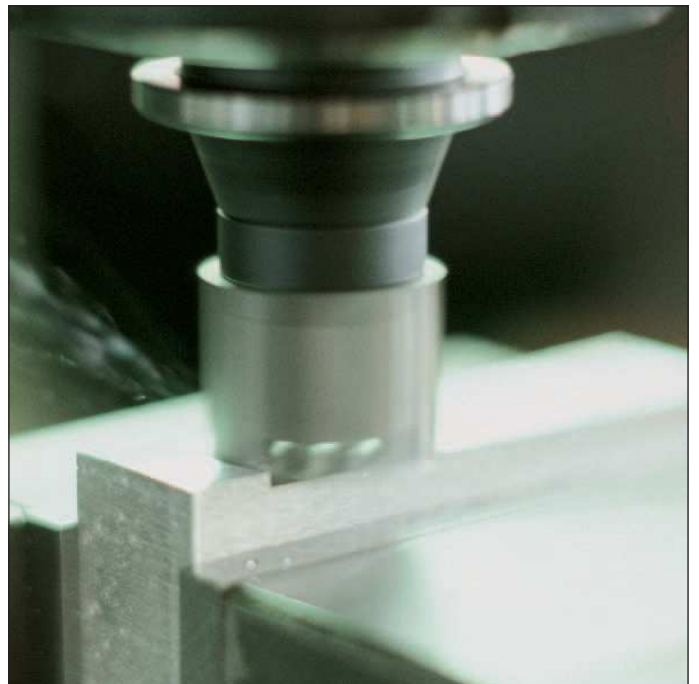
– Field-proven cycles for recurring operations

Fixed cycles for milling, drilling and boring

Frequently recurring operations that comprise several working steps are stored in the TNC 128 as cycles. You program them under conversational guidance and are supported by graphics that clearly illustrate the required input parameters.

Standard cycles

Besides the fixed cycles for drilling, tapping (with or without floating tap holder), face milling, rectangular pockets, rectangular studs, reaming and boring, there are also cycles for hole patterns and milling.



OEM cycles

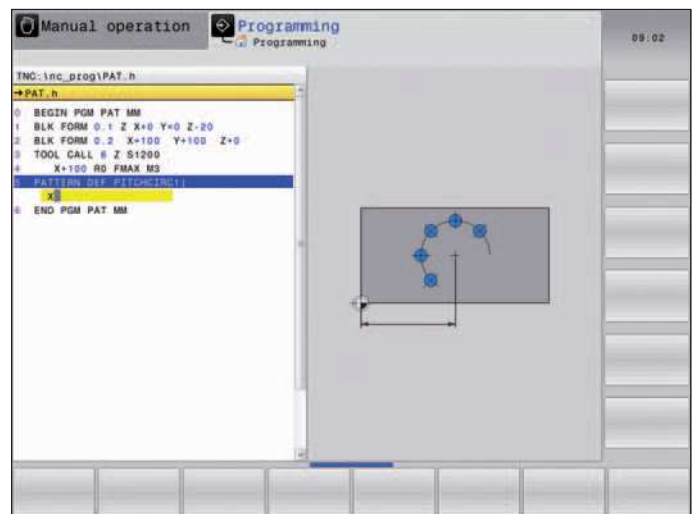
As original equipment manufacturers (OEMs), machine-tool builders can contribute their special manufacturing know-how by designing additional fixed cycles and saving them in the TNC 128. However, the end user can write his own cycles as well. HEIDENHAIN makes this possible with its PC program CycleDesign. This enables you to organize the input parameters and soft-key structure of the TNC 128 to suit your own needs.

Machining with parametric programming

Parametric programming also offers you a simple method of realizing operations for which no standard cycle is available. Here you can use the basic arithmetical operations, trigonometric functions, roots, powers, logarithmic functions, parentheses, and logical comparisons with conditional jump instructions.

Stay simple and flexible when programming machining patterns

Machining positions are often arranged in patterns on the workpiece. With the TNC 128, you can program very diverse machining patterns simply and extremely flexibly—of course with graphic support. You can define various point patterns with any numbers of points. Then you can execute all points at once or each point individually.



Programming in the workshop

– Reusing programmed contour elements

Coordinate transformation

If you should need a contour that has already been programmed at another position or in a different size, the TNC 128 offers you a simple solution: coordinate transformation.

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

Program section repeats and subprograms

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

In **program section repeats**, you label a section of the program and during program run the TNC repeats the section successively as many times as required.

You can mark a program section as a **subprogram** and then call it at any point in the program and as often as you want.

With the **program call** function you can even use a completely separate program at any place in your current program. This gives you convenient access to preprogrammed, frequently needed working steps or contours.

Of course you can also combine these programming techniques.



– Fast availability of all information

Do you have questions on a programming step but your User's Manual is not at hand? No problem: The TNC 128 now features TNCguide, a convenient help system that can show the user documentation in a separate window.

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

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

You can download the documentation in the desired language from the HEIDENHAIN homepage into the corresponding language directory on the TNC's memory medium.

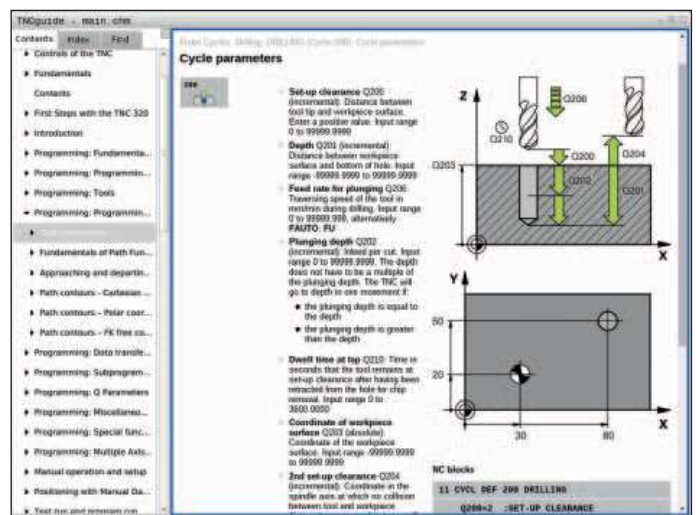
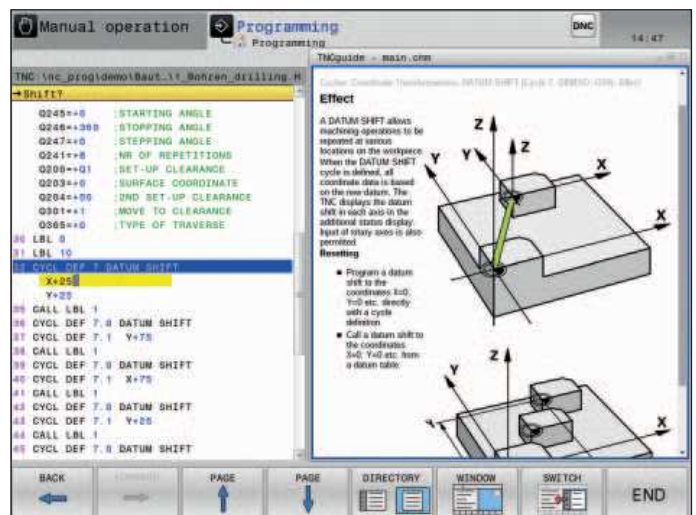
CAD viewer (standard)

The integrated CAD viewer allows you to open standardized 3-D CAD models and drawings directly on the TNC 128. This powerful viewer is a simple and simultaneously efficient solution for displaying CAD design data on the shop floor. Different viewing options and functions for rotating and zooming enable you to visually monitor and analyze your CAD data in detail.

Moreover, you can also use the viewer to find position values and dimensions from the 3-D model. And you can set the datum as desired and select elements in the model. The CAD viewer shows the coordinates of the selected elements in a window.

The TNC 128 can display the following file formats:

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



Open for communication

– Fast data transfer with the TNC

Archiving programs

For well-organized program management on your control, simply place the individual files in directories (folders). You can structure the respective directories through individual subdirectories.

The exchange of programs and data archiving have become a matter of course in modern production. If there is a production bottleneck on a machine, the program can simply be run on a machine with sufficient capacity available. No matter whether job orders repeat themselves, similar parts need to be produced, or reworking is necessary during repairs, the appropriate program is always at hand.

With the TNC 128 you are ideally prepared for such demands, since the control is easily connected to a PC or integrated in your network. Even in its standard version, the TNC 128 features a Gigabit Ethernet interface in addition to its RS-232-C/V.24 data interface.

USB interface

The TNC 128 also supports standard memory media with USB interface. You can use USB memory media to quickly and easily save programs and tool data, and exchange these with PCs or other machines.

Programs for data transfer

With the aid of the free PC software **TNCremo** from HEIDENHAIN and an Ethernet or other data interface you can

- transfer remotely stored part programs and tool tables in both directions and
- make backups.

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



Positioning with the electronic handwheel

– Delicate axis traverse

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

You can move the axis slide through the feed motors in direct relation to the rotation of the handwheel. For delicate operations you can set the transmission ratio to certain preset distances per handwheel revolution.

HR 130 panel-mounted handwheel

The panel-mounted handwheel from HEIDENHAIN can be integrated in the machine operating panel or mounted at another location on the machine.

HR 510 portable handwheel

If you need to get a closer look at the workpiece in the machine's working envelope, then ideally you should use the HR 510 portable handwheel. The axis keys and certain functional keys are integrated in the housing. In this way you can switch axes and set up the machine at any time—regardless of where you happen to be standing.

The following functions are available:

- Traverse direction keys
- Three keys with preset feed rates for latched traverse
- Actual-position-capture key
- Three keys for machine functions to be defined by the machine tool builder
- Permissive buttons
- Emergency stop button

HR 510



Workpiece measurement

– Setup, presetting and measuring with touch trigger probes

Workpiece touch probes from HEIDENHAIN help you to reduce costs: together with the TNC 128's cycles, touch probes can perform setup, measuring and inspection functions.

The stylus of a TS touch trigger probe is deflected upon contact with a workpiece surface. At that moment the TS generates a trigger signal that is transmitted via cable to the control.

The touch probe is inserted directly into the machine tool spindle. It can be equipped with various shanks depending on the machine. The ruby ball tips are available in several diameters, and the styli in different lengths.

The TNC 128 primarily uses touch probes with **signal transmission via cable** for machines with manual tool change:

TS 260
KT 130



* The touch probes must be interfaced to the TNC 128 by the machine tool builder.



KT 130



More information about workpiece touch probes is available on the Internet at www.heidenhain.de or in the Product Overview *Touch Probes – New Generation*.

Tool measurement

– Measuring length, radius and wear directly in the machine

The tool is of course a decisive factor in ensuring a consistently high level of production quality. This means that an exact measurement of the tool dimensions and periodic inspection of the tool for wear and breakage, as well as the shape of each tooth, are necessary. A suitable touch trigger probe for tool measurement is the **TT 160***. It is installed directly in the machine's workspace, where it permits tool measurement either before machining or during interruptions.

The TT 160 tool touch probe captures the tool length and radius. When probing the rotating or stationary tool, e.g. during individual tooth measurement, the contact plate is deflected and a trigger signal is transmitted directly to the TNC 128.

* The touch probe must be interfaced to the TNC 128 by the machine tool builder.



TT 160

More information about tool touch probes is available on the Internet at www.heidenhain.de or in the Product Overview *Touch Probes – New Generation*.

Overview

– User functions

User functions	Standard	Option
Short description	<ul style="list-style-type: none"> • <ul style="list-style-type: none"> ○ 1st additional axis for 4 axes plus open-loop or closed-loop spindle ○ 2nd additional axis for 5 axes and open-loop spindle 	
Program entry	<ul style="list-style-type: none"> • HEIDENHAIN conversational 	
Position entry	<ul style="list-style-type: none"> • Nominal positions for lines in Cartesian coordinates • Incremental or absolute dimensions • Display and entry in mm or inches 	
Tool tables	<ul style="list-style-type: none"> • Multiple tool tables with any number of tools 	
Cutting data	<ul style="list-style-type: none"> • Automatic calculation of spindle speed, cutting speed, feed per tooth and feed per revolution 	
Program jumps	<ul style="list-style-type: none"> • Subprograms • Program section repeats • Calling any program as a subprogram 	
Fixed cycles	<ul style="list-style-type: none"> • Cycles for drilling, pecking, reaming, boring, counterboring, conventional and rigid tapping • Multipass milling of smooth surfaces • Rectangular pockets • Face milling • Full-surface machining of rectangular pockets • Cartesian and polar point patterns • OEM cycles (special cycles developed by the machine tool builder) can be integrated 	
Coordinate conversions	<ul style="list-style-type: none"> • Shifting, mirroring, scaling (axis specific) 	
Q parameters Programming with variables	<ul style="list-style-type: none"> • Mathematical functions =, +, -, *, /, sin α, cos α, angle α of sin α and cos α, tan α, arc sin, arc cos, arc tan, a^n, e^n, ln, log, \sqrt{a}, $\sqrt{a^2 + b^2}$ • Logical operations (=, =, /, <, >) • Calculating with parentheses • Absolute value of a number, constant π, negation, truncation of digits before or after the decimal point • Functions for calculation of circles 	
Programming aids	<ul style="list-style-type: none"> • 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 128 • Graphic support for programming cycles • Comment and structure blocks in the NC program 	
Teach-In	<ul style="list-style-type: none"> • Actual positions can be transferred directly into the NC program 	

User functions	Standard	Option	
Test graphics Display modes	•	•	Graphic simulation before a program run, even while another program is running Plan view, projection in 3 planes, 3-D view, 3-D line graphic Detail zoom
Programming graphics	•		In the Programming and Editing mode, the contours of the NC blocks are drawn on screen while they are being entered (2-D pencil-trace graphics), even while another program is running
Program-run graphics Display modes	•	•	Graphic simulation during real-time machining Plan view/view in three planes/3-D view
Machining time	•	•	Calculation of machining time in the Test Run operating mode Display of the current machining time in the Program Run operating modes
Returning to the contour	•	•	Mid-program startup in any block in the program, returning the tool to the calculated nominal position to continue machining Program interruption, contour departure and return
Datum management	•		For saving any reference points
Datum tables	•		Multiple datum tables for storing workpiece-specific datums
Touch probe cycles	•	•	Calibrating the touch probe Set datum
Conversational languages	•		English, German, Czech, French, Italian, Spanish, Portuguese, Swedish, Danish, Finnish, Dutch, Polish, Hungarian, Russian (Cyrillic), Chinese (traditional, simplified), Slovenian, Slovak, Norwegian, Korean, Turkish, Romanian
CAD viewer	•		Display CAD models on the TNC

– Options, accessories

Option	Option number	As of NC software 771841-xx	ID	Comment
Additional axis	–	01	–	<ul style="list-style-type: none"> • 1st additional axis for 4 axes plus open-loop or closed-loop spindle • 2nd additional axis for 5 axes and open-loop spindle
Touch probe functions	17	01	634063-01	Touch probe cycles <ul style="list-style-type: none"> • Set datum • Tool measurement • Touch probe input enabled for non-HEIDENHAIN systems
HEIDENHAIN DNC	18	01	526451-01	Communication with external Windows applications over COM component
Python OEM process	46	01	579650-01	Python application on the TNC

Accessories	
Electronic handwheels	<ul style="list-style-type: none"> • One HR 510 portable handwheel or • one HR 130 panel-mounted handwheel, or • up to three HR 150 panel-mounted handwheels through an HRA 110 handwheel adapter
Workpiece measurement	<ul style="list-style-type: none"> • TS 260: Touch trigger probe with cable connection or • KT 130: Simple touch trigger probe with cable connection
Tool measurement	<ul style="list-style-type: none"> • TT 160: Touch trigger probe
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

Overview

– Specifications

Specifications	
Components	<ul style="list-style-type: none"> • Main computer with TNC keyboard and integrated 12.1-inch TFT color flat-panel display with soft keys
Operating system	<ul style="list-style-type: none"> • HEROS real-time operating system for machine control
Memory	<ul style="list-style-type: none"> • 1.8 GB (on CFR compact flash memory card)
Input resolution and display step	<ul style="list-style-type: none"> • Linear axes: To 1 μm • Angular axes: To 0.001°
Input range	<ul style="list-style-type: none"> • Maximum 99 999 9999 mm or 99 999 9999°
Block processing time	<ul style="list-style-type: none"> • 6 ms
Axis feedback control	<ul style="list-style-type: none"> • Position loop resolution: Signal period of the position encoder/1024 • Cycle time of position controller: 3 ms
Range of traverse	<ul style="list-style-type: none"> • Maximum 100 m
Spindle speed	<ul style="list-style-type: none"> • Maximum 100 000 rpm (analog speed command signal)
Error compensation	<ul style="list-style-type: none"> • Linear and nonlinear axis error, backlash, thermal expansion • Static friction
Data interfaces	<ul style="list-style-type: none"> • RS-232-C/V.24 max. 115 kbit/s • Extended data interface with LSV2 protocol for remote operation of the TNC 128 over the data interface with the HEIDENHAIN software TNCremo or TNCremoPlus • Gigabit Ethernet interface 1000BASE-T • 3 x USB (1 x front USB 2.0; 2 x back panel USB 3.0)
Diagnostics	<ul style="list-style-type: none"> • Fast and simple troubleshooting through integrated diagnostic aids
Ambient temperature	<ul style="list-style-type: none"> • Operation: +5 °C to +40 °C • Storage: –20 °C to +60 °C