

U

=>

Ð

X-

000

2 - N 8

.8.

IV+ Z+ Y+ V+ VI+

++

HEIDENHAIN TNC7 ه ⊙ م 02.Fans.Jangs.H x 95 m Import I Progen 📕 💽 💿 🔹 🗅 🕲 ") (* 😰 🖱 🙋 100 🔍 😔 🛛 × 🛛 I Smiller 📰 🔂 0 2 0 0 × ∩ € ġ, 120 - ROUGHING CIRCULAR STUD DODL CALL "MILL DOD, ROUGH" 2 STOOD 2+100 RB FUAX H3 YOL DEF 257 KMEISZAFFEN -0222++98 :FERTIGTEIL-DURCHE. -0222++141 :MORKPICE BLANK DIA. -0 ANCE FOR S D RATE MILL OR UP-OUT TIEFE -0 📲 5 D H HH 2 Katela. -Inset NC function 6010 te. Book scan Select in Program Run Start the ~ 8 2 3 Z A . -/+ В N X ∞ + a CE PEL P I APPR DEP

E -

GOTO --

PG DN



HEIDENHAIN



TNC7 basic

Overview of New and Modified Software Functions

NC Software 81762x-19

English (en) 09/2024

About this document

This document describes the new and modified software functions of the TNC7 basic. Each software version is covered in a separate chapter.

The chapters are structured as follows:

- New software options
- New functions
- Modified or extended functions

The contents of the subchapters are subdivided and sorted according to the chapters of the User's Manual. This makes it easier for you to find the desired information in the User's Manuals.

If a content is part of a software option, then the option number is indicated in parentheses.



User's Manual Complete Edition

All of the control's functions that are available to the user are described in the **Complete Edition** of the User's Manual.

The **Complete Edition** is available as a PDF for every software version.

ID: 1411730-xx

TNCguide

Have you found any errors or would you like to suggest changes?

We continuously strive to improve our documentation for you. Please help us by sending your suggestions to the following e-mail address:

tnc-userdoc@heidenhain.de

Table of contents

1	Software 81762x-19	7

Table of contents

1	Soft	ware 81	762x-19	7
	1.1	Modifie	ed or enhanced software options	8
		1.1.1	OPC UA NC Server (#56-61 / #3-02-1*)	8
	1.2	New fu	nctions	9
		1.2.1	About the product	9
		1.2.2	Status displays	9
		1.2.3	Programming fundamentals	9
		1.2.4	Path functions	9
		1.2.5	Programming techniques	10
		1.2.6	Files	10
		1.2.7	Text editor	10
		1.2.8	Collision monitoring	10
		1.2.9	Multiple-axis machining	11
		1.2.10	Programming with variables	11
		1.2.11	The Contour graphics workspace	11
		1.2.12	Opening CAD files with CAD Viewer	11
		1.2.13	ISO	12
		1.2.14	User aids	12
		1.2.15	The Simulation workspace	13
		1.2.16	Touch probes	13
		1.2.17	Touch probe functions in the Manual operating mode (#17 / #1-05-1)	13
		1.2.18	Touch-probe cycles for workpieces (#17 / #1-05-1)	14
		1.2.19	Program run	14
		1.2.20	Tables	14
		1.2.21	Electronic handwheel	14
		1.2.22	Integrated functional safety (FS)	14
		1.2.23	The Settings application	15
		1.2.24	Machine parameters	16
		1.2.25	User administration	17
		1.2.26	HEROS operating system	17
		1.2.27	Accessories	17

1.3	Modified or extended functions		18
	1.3.1	Status displays	18
	1.3.2	Programming fundamentals	18
	1.3.3	Tools	18
	1.3.4	Contour and point definitions	18
	1.3.5	Cycles for milling and drilling	19
	1.3.6	Programming techniques	19
	1.3.7	Files	20
	1.3.8	Text editor	20
	1.3.9	Collision monitoring	20
	1.3.10	Programming with variables	20
	1.3.11	User aids	21
	1.3.12	Touch probe functions in the Manual operating mode (#17 / #1-05-1)	21
	1.3.13	Touch-probe cycles for workpieces (#17 / #1-05-1)	21
	1.3.14	Pallet machining and job lists	21
	1.3.15	Program run	22
	1.3.16	Tables	22
	1.3.17	Electronic handwheel	22
	1.3.18	Integrated functional safety (FS)	23
	1.3.19	The Settings application	23
	1.3.20	Machine parameters	23
	1.3.21	User administration	23
	1.3.22	HEROS operating system	24



Software 81762x-19

1.1 Modified or enhanced software options

1.1.1 OPC UA NC Server (#56-61 / #3-02-1*)

Торіс	Description
Software option OPC UA NC Server (#56-61 / #3-02-1*)	On controls with SIK2 , you can enable up to ten (instead of six) OPC UA connections.

1.2 New functions

1.2.1 About the product

Торіс	Description
Remember position of windows	Use the Remember position icon to select whether the control will remember the position of a window the next time a window is opened.
Calculating numerical values during input	Use the +, -, *, /, (and) keys for calculations within numerical input fields and table cells.
Warning if not enough main memory is available	The control must have at least 16 GB of RAM, as the control will other- wise display a warning.

1.2.2 Status displays

Торіс	Description
The Status workspace	In the PGM tab of the Status workspace, you can open the Counter settings window in the Parts counter area. You can enter the current count and the target value for the counter.
	In the QPARA tab of the Status workspace, the Variables area has been added. In this area, the control shows the values of the named parameters you define.

1.2.3 Programming fundamentals

Торіс	Description
Insert NC function window	For some NC functions, the Insert NC function window offers the possibility of inserting the start and end of the NC function into the NC program at the same time (e.g., IF and END IF).
	If you mark several NC blocks in the NC program and insert the combined NC functions, the control will insert the corresponding NC function before and after the marked area.
Block scan Program run button	When you press the Block scan Program run button, the control opens the current file in the Program Run operating mode, as well as the Block scan window for the currently selected NC block.

1.2.4 Path functions

Торіс	Description
Programming a straight line L with actual position capture	Press the actual position capture key to program a straight line L with the actual positions of all defined axes. The control inserts the straight
	line L after the selected NC block.

9

1.2.5 Programming techniques

Торіс	Description
Control structures (e.g., with IF or ELSE)	The control provides the NC functions for programming the control structures.
	The control provides the following NC functions:
	IF, ELSE IF and ELSE case analyses
	FOR and WHILE program loops
	Expanded control of BREAK and CONTINUE program loops
	Using control structures, you can program the NC program more clear- ly and with a better structure. The control indents the NC blocks within the control structures. Thus you can see right away where a control structure starts and ends.

1.2.6 Files

Торіс	Description
Navigation path	In the file management, you can open a history of up to 20 previously used paths.
	You can edit the current navigation path.
User-defined filters	In the file management, you can define a user-defined filter for any desired file types. The filter is saved until you overwrite it.
The Open File workspace	If you have opened the Open File workspace in Editor operating mode and only tables are selected, the control displays the Use in simulation button.
	The Open File workspace opens a file preview that you can show or hide.

1.2.7 Text editor

Торіс	Description
The Text editor workspace	Editing functions (e.g., tab stops) have been added to the Text editor workspace.
	The Text editor workspace contains NC editor settings (e.g., to replace tab stops with spaces).
	In the Text editor workspace, you can open and edit any desired file types.

1.2.8 Collision monitoring

Торіс	Description
Set up fixtures (#140 / #5-03-2)	If the active unit of measure is inch, the control converts mm to inches within the Set up fixtures function.
Advanced checks	The Advanced checks function now includes the Collision between workpiece and machine check.
	The control displays a warning in case of collisions between the workpiece and the machine (e.g., spindle). The control does not consider the tool and the workpiece fixture.

1.2.9 Multiple-axis machining

Торіс	Description
Cylinder surface machining with CYLINDER SURFACE (#8 / #1-01-1)	The CYLINDER SURFACE NC function allows you to machine the cylinder surface with various NC functions, for example OCM cycles (#167 / #1-02-1), pocket milling cycles or path functions.

1.2.10 Programming with variables

Торіс	Description
Variable: named parameters	The control provides the named parameters variable type.
	The variable name of named parameters consists of a freely chosen designation enclosed in curly brackets (e.g., {DEPTH_1}).
	You can assign numerical and alphanumerical values to named parameters.
Format strings	The control provides the FMT syntax element for QS parameters and named parameters to define format strings. Using format strings, you no longer have to convert numerical values or concatenate strings. You can use formatted string parameters in the following NC functions: String formula SQL SELECT TEXT within FUNCTION REPORT
Q parameter list window	The machine manufacturer can define language-sensitive descriptive texts for variables. In the Q parameter list window, you can select between the standard descriptive text and the machine manufacturer's text or enter a descriptive text.

1.2.11 The Contour graphics workspace

Торіс	Description
Auto draw	In Editor mode, the control provides the Auto draw toggle switch.
	If you select an NC block and activate the toggle switch, the control will show a real-time graphic of the subsequently programmed contour in the Contour graphics workspace.
	If you mark several NC blocks and activate the toggle switch, the control will draw the contour of the marked NC blocks.

1.2.12 Opening CAD files with CAD Viewer

Торіс	Description
Rotate	The arrow icon in CAD Viewer now includes the Rotate mode. The Rotate mode is active by default and enables pure touch operation.
Output options	The Setting for whether comments are written to NC output files. icon has been added to CAD Viewer. Via this icon you can select whether CAD Viewer transfers information on the workpiece blank, the datum and the preset to the NC program

1.2.13 ISO

Торіс	Description
NC function G79 G00	The control calls the most recently programmed machining cycle at the position you defined in the NC block with G79 G00 . The control moves to the defined position at rapid traverse. G79 G00 corresponds to the Klartext syntax CYCL CALL POS with FMAX .
Syntax search	If the ISO editor toggle switch is active, you can search for identical syntax elements in different NC blocks.

1.2.14 User aids

Торіс	Description
Issuing notifications with FUNCTION REPORT	With the FUNCTION REPORT NC function, the controls issues notifications under program control. You can define the notification text yourself. If the machine manufacturer or another provider has saved notifications as a PO file, you can also output these notifications.
Search column in the Text editor workspace	In the Text editor workspace, the control provides the Search column. The search works in the same way as in the Program workspace.
Context menu in the Document workspace	In the Document workspace, the control provides a context menu with additional functions for every file type (e.g., navigating back within the opened files).
Showing NC blocks	You can show hidden NC blocks with the BACKSPACE key.
Calculator	 The calculator's keyboard provides the following input options: The P key corresponds to PI The RETURN or ENT keys correspond to = The DEL key corresponds to DEL
	When the actual position capture key is pressed, the control shows the current axis positions in the calculator. You can copy the current value of an axis into the calculator.
GOTO function in the Text editor workspace	In the Text editor workspace, use the GOTO record button to define the number of a line to be selected by the control.

1.2.15 The Simulation workspace

Торіс	Description
The Workpiece options column	The control provides the following functions also in the Manual and Program Run operating modes:
	Reset the workpiece
	Remove the chips
The Simulation settings window	The Simulation settings window is available in the Program Run and Manual operating modes. You can select whether the control will show a solid-model view.
	In the Tables area, the control shows the Reset button. With the Reset button, the control selects the same tables for the simulation that are active for program run.
Overview window with active simulation	If the simulation of another NC program is currently running, the control shows a window with the name of this NC program above the function bar. If you double-tap or click this window, the control switches from the active tab to the NC program currently being simulated.

1.2.16 Touch probes

Торіс	Description
Set Up Touch Probes	The Overview of touch probes menu item of the Settings application replaces the Set Up Touch Probes HEROS function.

You can set up touch probes using **TNCdiag**.

1.2.17 Touch probe functions in the Manual operating mode (#17 / #1-05-1)

Торіс	Description
Set up the workpiece (#159 / #1-07-1)	You can choose an NC program within the Set up the workpiece function. The control generates the 3D model from the workpiece blank definition of the NC program.
	If the active unit of measure is inch, the control converts mm to inches within the Set up the workpiece function.
	The contents and the representation of the error estimate diagram have been changed. The error estimate diagram shows for each touch point the distance of the touch point from the nominal position of the 3D model. The columns of the diagram are transparent until the status of all axes is green.
The Setup application	The control provides the Remove last measurement button to repeat an erroneous touch point, for example.

1.2.18 Touch-probe cycles for workpieces (#17 / #1-05-1)

Торіс	Description
Cycle 1403 RECTANGLE PROBING (#17 / #1-05-1)	This cycle determines the center, width and length of a rectangle. The control probes two opposing touch points in each case.
Cycle 485 MEASURE LATHE TOOL (#17 / #1-05-1)	This cycle enables you to measure lathe tools with a tool touch probe. You need a tool touch probe with a cuboid probe contact. This cycle can be executed only in FUNCTION MODE MILL machining mode.

1.2.19 Program run

Торіс	Description
Automatic program start	Use the Automatic program start function to define the moment from which the control autonomously executes an NC program.

1.2.20 Tables

Торіс	Description
The Table workspace	In the header of the Table workspace, the control may show the "Rules not met" filter icon, if applicable.
	The control displays only the rows that do not meet the rules defined by the machine manufacturer in CfgTableCellCheck (no. 141300).
Create new table window	In the Create new table window, you can choose between mm and inches as the unit of measure.

1.2.21 Electronic handwheel

Торіс	Description
HR 180 panel-mounted handwheel in the operating panel	The control supports the HR 180 panel-mounted handwheel.
Wireless handwheel	You can set up wireless handwheels using TNCdiag .

1.2.22 Integrated functional safety (FS)

Торіс	Description
F limited toggle switch	The F limited toggle switch has been removed.
Test status of the axes	You can reset the test status of individual axes or all axes in the Axis status menu item of the Settings application.
	To reset the test status of axes, you need the NC.ApproveFsAxis right. This right is only available if user administration is active.

1.2.23 The Settings application

Торіс	Description
Menu item Adjustment of analog voltage offset	In the Adjustment of analog voltage offset menu item, the control shows all connected analog axes, whether the axes are in closed-loop control, and the current voltage offset. If the axis permits it, you can change the value of the voltage offset.
SIK menu item	In the SIK Information area, the control shows whether it features SIK or SIK2 .
DNC menu item	The Fingerprint of the host key function has been added. If you press the Show button, the control shows a unique ASCII graphic, comparable to a fingerprint. If you configure a secure connection, you can compare this ASCII graphic with a graphic included in the client application. That way you can ensure the connection is set up with the correct control.
OPC UA NC Server (#56-61 / #3-02-1*)	The machine manufacturer can permit login with a user name and password, for example for client applications that do not support login with a user certificate.
	In the OPC UA menu item of the Settings application, the control indicates the options available to the current user for logging in.
	Client applications can change the counter reading with the aid of the OPC UA and the NC.RemoteOperator role.

1.2.24 Machine parameters

Торіс	Description
Configuration editor	In the Configuration window, the Display in tree view toggle switch has been added. With this toggle switch, you can display the machine parameters in a tree view instead of the structure view.
	The Go to number button has been added; you can use this button to enter the number of a machine parameter and move directly to this parameter.
	In the configuration editor, you can insert objects with the key combina- tion CTRL + N .
	If the table view of the configuration editor is active, you can change the width of the active column via the Change column width icon.
SYM (SEQ) tilting solution	Via the optional machine parameter prohibitSEQ (no. 201209), the machine manufacturer defines whether only SYM or also SEQ are available for selection.
Reactions of component monitoring (#155 / #5-02-1)	Use the machine parameter enforcedReactions (no. 129403) to define the component monitoring reactions to be carried out by the control. By default, the control carries out all reactions.
	This machine parameter replaces the previous machine parameters enforceReaction (no. 129401) and showWarning (no. 129402).
Conversational language	The control supports Japanese as a conversational language. The desired conversational language can be selected via machine parameters ncLanguage (no. 101301) and plcDialogLanguage (no. 101302).
	Use the optional machine parameter noRebootDialog (no. 101306) to define whether, after a change of the conversational language, the control displays a restart message.
Override controller	Via the optional machine parameter ocColourScheme (no. 103413), the machine manufacturer defines an alternative color scheme for the override controller.

1.2.25 User administration

Торіс	Description
Rights	The HEROS.NormalUser role includes the HEROS.MountUSBDevices right. You need this right to be able to connect a USB device to the control.
	The control provides the NC.OpmodeSingleStep right, which allows you to execute NC programs in Single Block mode.
	The control provides the NC.OpcUaPwAuth right. With this right and a user name and password, you can log in to the OPC UA NC Server (#56-61 / #3-02-1*).
	The control provides the NC.OpcUaPwAuthOnlyMachineNet right. With this right and a user name and password, you can log in to the OPC UA NC Server (#56-61 / #3-02-1*) via the eth1 network interface.
Machine parameters	The machine manufacturer defines which machine parameters are saved user-specifically by the control when user administration is active. These machine parameters can be changed at any time without, for example, having to restart the control.
Buttons	When user administration is active, the control provides the Lock screen, Change user and Log off user buttons in the Start/Login application.
Function users pre-defined by the machine manufacturer	HEIDENHAIN provides templates for the oemreadonly and oemautomation function users, which the machine manufacturer can activate. These function users can be used to set up and operate exter- nal systems (e.g., robots). Refer to your machine manual.

1.2.26 HEROS operating system

Торіс	Description
HEROS menu	The Hostkey HEROS tool has been added; the control can use this tool to display a unique ASCII graphic, similar to a fingerprint. If you config- ure a secure connection, you can compare this ASCII graphic with a graphic included in the client application. That way you can ensure the connection is set up with the correct control.

1.2.27 Accessories

Торіс	Description
ITC	Using the Batch Process Manager BPM (#154 / #2-05-1), you can process pallet tables on the ITC.
Handwheel	The HRA 180 handwheel adapter and the HR 180 panel-mounted handwheel replace the HRA 110 and HR 150 products.

1.3 Modified or extended functions

1.3.1 Status displays

Торіс	Description
The Status workspace	The MON tab of the Status workspace has been renamed as COMPMON (#155 / #5-02-1).
	The machine manufacturer defines the contents to be shown in the COMPMON tab of the Status workspace (#155 / #5-02-1).
	If the values of Cycle 32 TOLERANCE are restricted by Dynamic Collision Monitoring DCM (#40 / #5-03-1), the control will show the text DCM limited in the CYC tab of the Status workspace after the value concerned.
Display of the axis names	If the name of an axis has more than two characters, the control will adapt the width of the orange background.
Program runtime in the status overview on the TNC bar	If the TNC bar is minimized, the control indicates the program runtime with the units ${f m}$ and ${f s}$ or ${f h}$ and ${f m}$.

1.3.2 Programming fundamentals

Торіс	Description
Marking an area using identi- cal syntax elements	If you press SHIFT + UP or SHIFT + DOWN during editing, the control will jump to the next NC block with the same syntax element. In doing so, the control marks the two NC blocks and the area in between.
Text editor mode	The Autocomplete in text mode function additionally offers the possibility of choosing a tool via the selection dialog, for example.
	Using the arrow keys to the left and right, you can transfer syntax elements from the selection menu of the autocomplete feature to the NC program.

1.3.3 Tools

Торіс	Description
Preselecting a tool with TOOL DEF	You can no longer program L and R in the TOOL DEF NC function. The control issues an error message during execution.

1.3.4 Contour and point definitions

Торіс	Description
PATTERN DEF	The control shows a matching icon for the selection options of the PATTERN DEF NC function.
Support for *.hp point files	The control no longer supports point files with the extension *.hp Up to and including software version 18, the control converted point files with the extension *.hp . During execution, the control automatical- ly generated a file with the extension *.hp.pnt.dep . You can also use this file with software version 19.

1.3.5 Cycles for milling and drilling

Торіс	Description
Cycle 24 SIDE FINISHING (ISO: G124)	If the sum of the finishing allowance for the side Q14 and the radius of the finishing mill is smaller than the sum of the finishing allowance for the side Q3 and the radius of the roughing mill, the control no longer displays an error message.
	This allows you to perform finishing tasks with a tool that is only slight- ly larger than the roughing tool.
Cycle 32 TOLERANCE (ISO: G62)	The T-FMAX parameter has been added to Cycle 32 TOLERANCE . This parameter defines a tolerance for rapid-traverse movements.
Cycle 224 DATAMATRIX CODE PATTERN (ISO: G224)	The following parameters have been added to Cycle 224 DATAMATRIX CODE PATTERN :
	 Q661 SYMBOL SIZE: number of rows and columns of the pattern Q367 CODE POSITION: position of the starting point relative to the pattern
Cycle 225 ENGRAVING (ISO: G225)	The special characters €, ° and © have been added to Cycle 225 ENGRAVING.
Cycle 274 OCM FINISHING SIDE (ISO: G274) (#167 / #1-02-1)	 The behavior of Cycle 274 OCM FINISHING SIDE has been modified: With Q338=0 INFEED FOR FINISHING, the control performs finishing with as few downfeeds as possible. If the contour contains, for example, several islands with different heights, the control no longer machines each height individually, but rather starts at the maximum depth possible. Thus, the control needs fewer infeeds and can reduce the machining time. If the sum of the finishing allowance for the side Q14 and the radius of the finishing mill is smaller than the sum of the finishing allowance for the roughing mill, the control no longer displays an error message. This allows you to perform finishing tasks with a tool that is only slightly larger than the roughing tool
Cycle 277 OCM CHAMFERING (ISO: G277) (#167 / #1-02-1)	The parameter Q240 NUMBER OF CUTS has been added to Cycle 277 OCM CHAMFERING . This parameter allows you to program chamfer- ing in several cuts. The depth of the tool tip remains constant during the individual cuts, the control performs a lateral infeed. The control distributes the cuts evenly to attain a constant chip cross section over all infeeds.
OCM cutting data calculator (#167 / #1-02-1)	The material database for the OCM cutting data calculator now contains additional steels with U.S. designations.

1.3.6 Programming techniques

Торіс	Description
NC sequences	You can save up to 2000 sequential NC blocks as one NC component.
	You can save user-defined folder icons for the subfolders of the NC components.
CALL LBL	In the selection menu of CALL LBL , the control shows the comment in addition to the number or name of the label.

1.3.7 Files

Торіс	Description
The Update TAB / PGM function	The control supports only tables with UTF-8 character encoding. With the Update TAB / PGM function, the control changes the character encoding to UTF-8, if applicable.
Sequence of the file informa- tion	The control displays the file information in the sequence: date, time and file size.
The Quick selection new table workspace	In the Active tables for simulation area, you can open the tool tables selected in the Simulation settings window as tabs in the Tables operating mode.

1.3.8 Text editor

Торіс	Description
The Text editor workspace	Pressing the ENT key inserts a line break in the Text editor workspace.
Note regarding external file modifications	If the currently opened file was modified by another editor, the control will update the file content and display a note.

1.3.9 Collision monitoring

Торіс	Description
Set up fixtures (#140 / #5-03-2)	The contents and the representation of the error estimate diagram have been changed. The error estimate diagram shows for each touch point the distance of the touch point from the nominal position of the 3D model. The columns of the diagram are transparent until the status of all axes is green.
Combining fixtures	The individual components of a combined fixture retain assigned attrib- utes (e.g., colors).
Advanced checks	The Workpiece collision check has been renamed to Collision between workpiece and tool.

1.3.10 Programming with variables

Торіс	Description
Q parameter list window	Via the Global search toggle switch you can choose whether the control searches all columns of the Q parameter list window or only the currently selected window.
FN 18: SYSREAD (ISO: D18)	If you read the data of the current tool with FN 18: SYSREAD (ISO: D18) (e.g., ID950), the control shows the data of the new tool right from the start of the tool change.

1.3.11 User aids

Торіс	Description
The Search column	The search also considers space characters at the start of the search term.
The Structure column	The control shows ISO cycles in the Structure column.
Selecting text with touch operation	When selecting text during touch operation, the control will show two selection symbols below the text. With these symbols you can change the marked area by dragging.

1.3.12 Touch probe functions in the Manual operating mode (#17 / #1-05-1)

Торіс	Description
Working plane is inconsis- tent! (#8 / #1-01-1) window	The 3-D ROT Apply status function no longer transfers only the current positions of the rotary axes to the 3-D rotation window. For a consistent machining plane, the control takes over the status of the tilting function and the spatial angle from the Program Run operating mode or the MDI application, if necessary.
Set up the workpiece (#159 / #1-07-1)	By default, the control positions the 3D model to the active preset.
	If the active preset contains at least one spatial angle, the control by default selects the 6D probing mode.
Change the preset icon	The position of the Change the preset icon in the probing functions has been modified. When you select a touch probe function, the control immediately displays a message for checking the preset.

1.3.13 Touch-probe cycles for workpieces (#17 / #1-05-1)

Торіс	Description
Touch probe cycles 42x and 43x (#17 / #1-05-1)	The control saves the status of the message in parameters Q180 to Q182 before it issues the measurement report. If you interrupt machining by issuing a measurement report on the screen, you can determine the status of the measurement and stop machining, if necessary.
Cycle 1404 PROBE SLOT/ RIDGE (ISO: G1404) (#17 / #1-05-1)	You can combine Cycle 1404 PROBE SLOT/RIDGE with Cycle 1493 EXTRUSION PROBING . This can be used, for example, to detect any shape deviations.
Machine parameters	In the optional machine parameter trackAsync (no. 122503), the machine manufacturer defines whether the control orients the spindle for probing during prepositioning. This can save time during automatic probing processes. In addition, the control takes the calibrated center offset of L-shaped styli into account for the spindle tracking speed. This means that the speed at the ball tip is at most the rapid traverse of probe FMAX , which increases safety during probing.

1.3.14 Pallet machining and job lists

Торіс	Description
Editing the pallet table	Although a pallet table has been selected in Program Run mode of operation, you are able to edit in Editor mode.

1.3.15 Program run

Торіс	Description
Block scan	If program run is canceled during a program section repeat or program loop, the control provides the number of the repeat as point of interrup- tion.
	If the machine parameter operatingTimeReset (no. 200801) is defined with the value TRUE , the effect has changed. The control no longer resets the program run time when you start block scan to the point of interruption after a program cancellation.
Open in the editor button	In Editor operating mode, the control selects the same NC block that is currently selected in Program Run operating mode.

1.3.16 Tables

Торіс	Description
Table filters	The control displays user-defined filters under the All filter. You can select and deselect the user-defined filters.
	When you tap or click a filter once, the control activates only the select- ed filter in the corresponding area. When you double-tap or click a filter, the control activates the selected filter in addition to the active filters.
The Form workspace	The control displays the contents of some tables in groups within the Form workspace. Contents that are not assigned to any group are displayed under Not categorized . In the Tool_management application, for example, the area contains tool parameters that are irrelevant to the current tool type.
Characteristics of the table cells	The machine manufacturer can define the color and font of table cells.
The TABDATA function	In the TABDATA functions, you can also enter the table row as a number or numerical parameter.
The Filter column	For tables without default filters, the control displays the Filter column as soon as a user-defined filter is saved.
Machine manufacturer settings	In the optional machine parameter choice (no. 105704), the machine manufacturer can define toggle switches of the Form workspace. The machine manufacturer can change the icon and adapt the background color.
The Pocket table application	The Reset row button has been removed from the Pocket table application.

1.3.17 Electronic handwheel

Торіс	Description
Rotational speed in the display handwheel	The display of the handwheel always shows the spindle speed of the current channel (for example for a machine with several tool spindles).
Wireless handwheel	The control displays a warning if you connect a wireless handwheel with an already selected radio channel.

1.3.18 Integrated functional safety (FS)

Торіс	Description
Self-test of the control	If the self-test of the control is active, the control displays an icon in the information bar.
Internally monitored axes	Internally monitored axes can be activated and deactivated at run-time (e.g., interchangeable heads). The machine manufacturer must config- ure the activation and deactivation.
Check axis positions	In the Referencing workspace, you can switch as desired between the Referencing and Check axis positions modes.

1.3.19 The Settings application

Торіс	Description
PKI Admin	The PKI Admin has been reorganized. The Advanced settings tab has been removed and the settings were transferred to the appropriate tabs.
The VNC menu item	If a VNC connection is active, the control shows the icon of the connec- tion status in the information bar.
TNCscope	The TNCscope application can only be opened with machine manufac- turer rights.

1.3.20 Machine parameters

Торіс	Description
Configuration editor	The key combination CTRL + F opens the Search column in the config- uration editor.
	No longer needed machine parameters of the TNC 640 have been removed from the configuration editor of the TNC7 basic.

1.3.21 User administration

Торіс	Description
Connection to Windows domain	To join a Windows domain, the IT administrator must set up a function user. You can no longer join a Windows domain with a computer account.
	The control no longer automatically checks for every connection whether all of the required roles in the domain have been defined. Press the Check missing role definitions button to start the check.
Rights	The NC.OpmodeProgramRun right now only comprises Program run in Full Sequence mode, no longer in Single Block mode.
Function users pre-defined by the machine manufacturer	The maximum number of function users has been increased from 16 to 32.

1.3.22 HEROS operating system

Торіс	Description
Firewall	The firewall has been revised. You can protect every interface and source with the firewall.
HEROS menu	The TNCscope application can only be opened with machine manufac- turer rights.

HEIDENHAIN

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

Technical supportImage: 149 8669 32-1000Measuring systemsImage: 149 8669 31-3104service.ms-support@heidenhain.deNC supportImage: 149 8669 31-3101service.nc-support@heidenhain.deNC programmingImage: 149 8669 31-3103service.nc-pgm@heidenhain.dePLC programmingImage: 149 8669 31-3102service.plc@heidenhain.deAPP programmingImage: 149 8669 31-3102service.plc@heidenhain.deAPP programmingImage: 149 8669 31-3106service.app@heidenhain.de

www.heidenhain.com