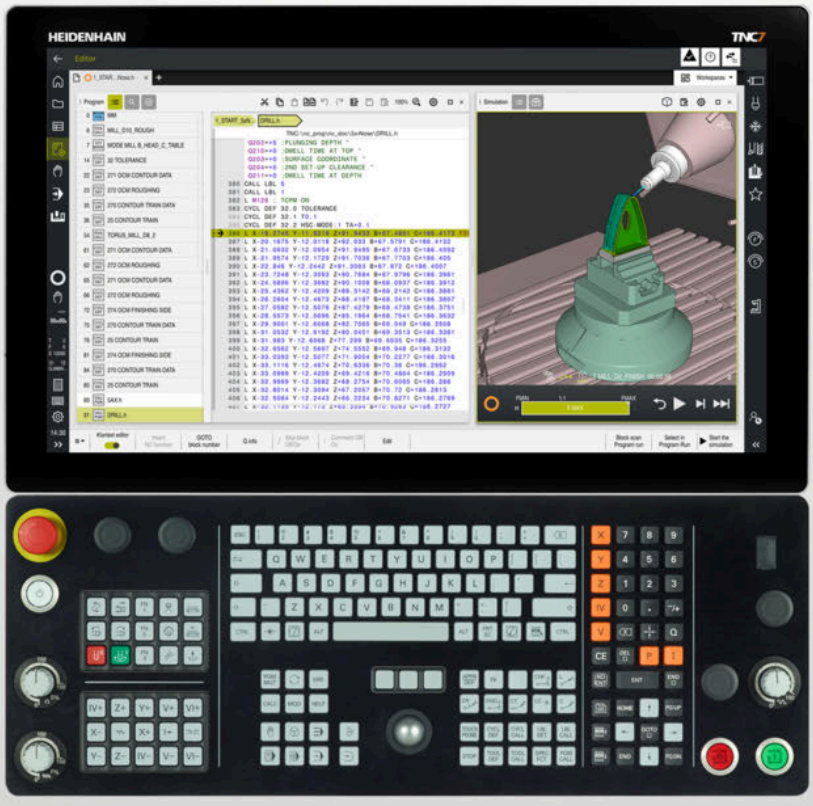
HEIDENHAIN

TNC7

Overview of New and Modified Software Functions



NC Software
81762x-17 to 81762x-19

English (en)
09/2024

About this document

This document describes the new and modified software functions of the TNC7. 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: 136999-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-17

1.1 New software options

1.1.1 Model Aided Setup (option 159)

Topic	Description
Software option 159: Model Aided Setup	This software option is used to determine the position and misalignment of a workpiece with only one touch-probe function. Complex workpieces with, for example, free-form surfaces or undercuts can be probed; this is sometimes not possible with the other touch-probe functions. The software option includes the **Set up the workpiece** touch probe function.

1.2 New functions

1.2.1 Programming fundamentals

Topic	Description
Text editor	In the text editor mode, the control provides an auto-complete function when programming. The control suggests syntax elements matching your entries which you can apply to the NC program.
	If an NC block contains a syntax error, the control displays a symbol in front of the block number. When you select the symbol, the control displays the corresponding error description.
Depiction of the NC program	If the control does not process or simulate the miscellaneous function **M1** or NC blocks hidden with **/**, it then grays out the miscellaneous function or NC blocks.
The **Program settings** window	In the **Klartext** area of the **Program settings** window, you select whether the control skips the offered optional syntax elements of an NC block during input. If the toggle switches in the **Klartext** area are active, the control skips the syntax elements Comment, Tool index and Linear superimposition.

1.2.2 Tools

Topic	Description
Tool types	The following tool types have been added: ■ **Face mill** (**MILL_FACE**) ■ **Chamfer cutter** (**MILL_CHAMFER**)
Tool table	You define a database ID for the tool in the DB_ID column of the tool table. In a tool database for all machines, you can identify tools with unique database IDs (e.g., within one machine shop). This allows you to coordinate the tools of multiple machines more easily.
	You define a radius at the tip of the tool in the **R_TIP** column of the tool table.
Touch probe table	You define the shape of the stylus in the **STYLUS** column of the touch probe table. You define an L-shaped stylus with the **L-TYPE** selection.
Grinding tool table (option 156)	You define the compensation method for dressing in the **COR_TYPE** input parameter for grinding tools (option 156): ■ **Grinding wheel with compensation, COR_TYPE_GRINDTOOL** Stock removal on the grinding tool ■ **Dressing tool with wear, COR_TYPE_DRESSTOOL** Stock removal on dressing tool

1.2.3 Path functions

Topic	Description
Superimpositioning on circular paths	When programming circular paths with **C**, **CR** and **CT**, the **LIN_** syntax element is now available in order to superimpose a linear motion over the circular motion of an axis. This allows you to program a helix in a simple way. In ISO programs, you can define a third axis in conjunction with the **G02**, **G03**, and **G05** functions.

1.2.4 Programming techniques

Topic	Description
NC sequences	You can save up to 200 successive NC blocks as NC sequences and insert them during programming using the **Insert NC function** window. In contrast to the called NC programs, you can adapt the NC sequences after insertion without changing the actual sequence.

1.2.5 Files

Topic	Description
Document workspace	The **Document** workspace has been added. In the **Document** workspace, you can open files in order to view them, such as a technical drawing.

1.2.6 Programming with variables

Topic	Description

FN 18: SYSREAD (ISO: **D18**)	The **FN 18: SYSREAD** (ISO: **D18**) functions have been enhanced:
	■ **FN 18: SYSREAD (D18) ID610 NR49**: Mode of filter reduction of one axis (**IDX**) for **M120**
	■ **FN 18: SYSREAD (D18) ID780**: Information on the current grinding tool
	  ■ **NR60**: Active compensation method in **COR_TYPE** column
	  ■ **NR61**: Inclination angle of dressing tool
	■ **FN 18: SYSREAD (D18) ID950 NR48**: Value in column **R_TIP** in the tool table for the current tool
	■ **FN 18: SYSREAD (D18) ID11031 NR101**: File name of the log file of Cycle **238 MEASURE MACHINE STATUS**

1.2.7 ISO

Topic	Description
ISO programs	You can run and edit ISO programs.

1.2.8 User aids

Topic	Description
Context menu	In the context menu of the **Editor** operating mode and the **MDI** application, the control offers the **Insert last NC block** function. With this function you can insert the last deleted or edited NC block in any NC program.

1.2.9 Simulation workspace

Topic	Description
Clamping situation	In the **Visualization options** column of the **Simulation** workspace, you can use the **Clamping situation** toggle switch to show the worktable and, if applicable, the fixtures in **Workpiece** mode.

1.2.10 Programmable touch probe cycles

Topic	Description
Cycle **1416 INTERSECTION PROBING** (ISO: **G1416**)	This cycle allows you to determine the intersection of two edges. The cycle requires a total of four touch points and two positions per edge. You can use the cycle in the three object planes **XY**, **XZ** and **YZ**.
Cycle **1404 PROBE SLOT/ RIDGE** (ISO: **G1404**)	This cycle determines the center and the width of a slot or ridge. The control probes two opposing touch points. You can also define a rotation for the slot or the ridge.
Cycle **1430 PROBE POSITION OF UNDERCUT** (ISO: **G1430**)	This cycle determines a single position with an L-shaped stylus. The control can probe undercuts due to the shape of the stylus.
Cycle **1434 PROBE SLOT/ RIDGE UNDERCUT** (ISO: **G1434**)	This cycle determines the center and the width of a slot or ridge with an L-shaped stylus. The control can probe undercuts due to the shape of the stylus. The control probes two opposing touch points.

1.2.11 Program run

Topic	Description
Navigation path	If you execute an NC program or a pallet table or if you test it in the opened **Simulation** workspace, the control displays a navigation path in the file information bar of the **Program** workspace. The control displays the names of all the NC programs used in the navigation path and opens the contents of all NC programs in the workspace. This makes it easier to keep an overview of the execution when calling programs and allows navigating between the NC programs when the program run is interrupted.

1.2.12 Tables

Topic	Description
Form workspace	The control displays an icon of the selected tool type in the **Tool Icon** area. For the turning tools the icons also take into account the tool orientation and show where the relevant tool data will apply.
	Use the up and down arrows in the title bar to select the previous or next table row.
Filtering tables	You can create user-defined filters for the tool tables and pocket table. To do this, define a search condition in the **Search** column which you save as a filter.
Importing tables	You can transfer tables from earlier control models to the TNC7. If columns are missing in the table, the control opens the **Incomplete table layout** window.

1.2.13 Settings application

Topic	Description
Update documentation	You can use the **Update documentation** function to install or update, for example, the **TNCguide** integrated product aid.
Configurations	Each user can create and activate configurations in which the control's user interface is customized. You can save and activate custom modifications to the control's user interface as a configuration, e.g. for each operator. The configuration contains, for example, favorites and the arrangement of the workspaces.
OPC UA NC Server (options 56 to 61)	The **OPC UA NC Server** enables client applications to access the tool data of the control. You can read and write tool data. The **OPC UA NC Server** does not provide access to the grinding and dressing tool tables (option 156).

1.2.14 Machine parameters

Topic	Description
Help graphics	Use the machine parameter **stdTNChelp** (no. 105405) to define whether the control displays help graphics as pop-up windows in the **Program** workspace.
The **Handwheel superimp.** function (option 44)	The optional machine parameter **CfgGlobalSettings** (no. 128700) allows you to define whether the control offers the parallel axes for **Handwheel superimp.**.

1.3 Modified or extended functions

1.3.1 Operation

Topic	Description
Window	You can change the size of windows. The control remembers the size until it is shut down.
Applications	In the **Files**, **Tables** and **Editor** operating modes, a maximum of ten tabs can be open at the same time. If you try to open additional tabs, the control shows a message.

1.3.2 Accessories

Topic	Description
Additional operating station	The control no longer supports the ITC 750 additional operating station.

1.3.3 Status displays

Topic	Description
Status overview on the TNC bar	In the status overview, the control displays the run time of the NC program in mm:ss format. As soon as the run time of the NC program exceeds 59:59, the control shows the run time in hh:mm format.
	If a tool usage file is available, the control calculates for the **Program Run** operating mode how long the execution of the active NC program will take. During program run the control updates the remaining run time. The control shows the remaining run time in the status overview on the TNC bar.
	If more than eight axes are defined, the control shows the axes in two columns in the position display of the status overview. With more than 16 axes, the control shows the axes in three columns.
Feed rate limitation	If a feed rate limit is active, the control highlights the **FMAX** button in color and displays the defined value. In the **Positions** and **Status** workspaces, the control shows the feed rate in orange.
	If the feed rate is limited using the **F LIMIT** button, the control displays **LIMIT** in square brackets.
	If the feed rate is limited by means of functional safety (FS), then the control displays the active safety function in brackets.
Status workspace	The **TRANS** tab of the **Status** workspace indicates the active shift in the working plane coordinate system **WPL-CS**. If the shift comes from a compensation table (***.WCO**), the control shows the path to the compensation table as well as the number and, if applicable, the comment of the active row.
	In the **Tool** tab of the **Status** workspace, the control displays the values of the **Tool geometry** and **Tool allowances** areas with four instead of three decimal places.
Handwheel	If a handwheel is active, the control shows the contouring feed rate in the display during program run. If only the currently selected axis is moving, the control shows the axis feed rate.

1.3.4 Powering on and off

Topic	Description
Shut down	If you shut down the control with still unsaved changes in NC programs and contours, the control displays the **Close the program** window. You can save the changes, discard them or cancel the shutdown.

1.3.5 Programming fundamentals

Topic	Description
Input	When you save an input value, the control removes superfluous zeros at the beginning of the input and at the end of the decimal places. The input range must not be exceeded for this.
	The control no longer interprets tab characters as syntax errors. In comments and structure items, the control displays a tab character as a space. In syntax elements, the control removes a tab character.
	If you edit a value and press the backspace key, the control deletes only the last character and not the complete input.
The **Insert NC function** window	If software options are not enabled, the control shows unavailable contents in the **Insert NC function** window grayed out.
	In the areas **Search result**, **Favorites** and **Last functions**, the control shows the path of the NC functions.
	If you select an NC function and swipe to the right, the control displays the following file functions: ■ Add to or remove from favorites ■ Open containing folder Only when you search for an NC function
Text editor	You can delete an empty line with the backspace key in text editor mode.

1.3.6 Tools

Topic	Description
Tool call	If you select the tool with the selection window when calling the tool with **TOOL CALL**, you can switch via an icon to the **Tables** operating mode. In this case, the control displays the selected tool in the **Tool management** application.
Touch probe table	The minimum input value of the **FMAX** column in the touch probe table has been changed from –9999 to +10.
Tool table	The maximum input range of the **LTOL** and **RTOL** columns of the tool table has been increased. It was from 0 to 0.9999 mm, and is now from 0.0000 to 5.0000 mm.
	The maximum input range of the **LBREAK** and **RBREAK** columns of the tool table has been increased. It was from 0 to 3.2767 mm, and is now from 0.0000 to 9.0000 mm.
	You can import tool tables of the TNC 640 as CSV files.
Tool test	If you double tap or click a tool in the **Tool check** column of the **Program** workspace, the control switches to the **Tables** operating mode. In this case, the control displays the selected tool in the **Tool management** application.

1.3.7 Path functions

Topic	Description
Line **L**	If you press the **actual position capture** key in the **Editor** operating mode or the **MDI** application, the control creates a straight line **L** with the current position of all axes.

1.3.8 Machining cycles

Topic	Description
Cycle **19 WORKING PLANE** (ISO: **G80**, option 8)	You can edit and execute Cycle **19 WORKING PLANE** (ISO: **G80**, option 8), but you cannot insert it into an NC program as a new element.
Cycle **277 OCM CHAMFERING** (ISO: **G277**, option 167)	Cycle **277 OCM CHAMFERING** (ISO: **G277**, option 167) monitors contour damage on the floor caused by the tool tip. This tool tip results from the radius **R**, the radius at the tool tip **R_TIP**, and the point angle **T-ANGLE**.
Cycle **292 CONTOUR.TURNG.INTRP.** (ISO: **G292**, option 96)	The parameter **Q592 TYPE OF DIMENSION** has been added to Cycle **292 CONTOUR.TURNG.INTRP.** (ISO: **G292**, option 96). This parameter is used to define whether the contour is programmed with radius dimensions or diameter dimensions.
M109 and **M110**	The following cycles consider the miscellaneous functions **M109** and **M110**: ■ Cycle **22 ROUGH-OUT** (ISO: G122) ■ Cycle **23 FLOOR FINISHING** (ISO: G123) ■ Cycle **24 SIDE FINISHING** (ISO: G124) ■ Cycle **25 CONTOUR TRAIN** (ISO: G125) ■ Cycle **275 TROCHOIDAL SLOT** (ISO: G275) ■ Cycle **276 THREE-D CONT. TRAIN** (ISO: G276) ■ Cycle **274 OCM FINISHING SIDE** (ISO: G274, option 167) ■ Cycle **277 OCM CHAMFERING** (ISO: G277, option 167) ■ Cycle **1025 GRINDING CONTOUR** (ISO: G1025, option 156)

1.3.9 Coordinate transformation

Topic	Description
The **3-D rotation** window (option 8)	In the **3-D rotation** window (option 8), if you enable a function in the **Manual Operation** or **Program run** areas, the control highlights the area in green.

1.3.10 Compensations

Topic	Description
FUNCTION PROG PATH (option 9)	If you define a grinding tool (option 156) with orientation **9** or **10**, the control supports circumferential milling in conjunction with **FUNCTION PROG PATH IS CONTOUR** (option 9).

1.3.11 Files

Topic	Description
File management	The control shows the occupied memory and total memory of the drives in the navigation bar of the file management.
	The control shows STEP files in the preview area.
	When you cut a file or folder in the file management, the control grays out the icon of the file or folder.
	When you add a favorite or lock a file in the file management, the control displays an icon next to the file or folder.
Quick selection workspace	Tables for execution and simulation can be opened in the **Quick selection** workspace in the **Tables** operating mode.
	In the **Quick selection** workspace in the **Editor** operating mode, you can create NC programs with mm or inch units of measurement as well as ISO programs.

1.3.12 Programming with variables

Topic	Description
FN 16: F-PRINT (ISO: **D16**)	For a screen output with **FN 16: F-PRINT** (ISO: **D16**), the control displays a pop-up window.
The **Q parameter list** window	The window **Q parameter list** contains an input field that allows you to navigate to a unique variable number. If you press the **GOTO** key, the control selects the input field.

1.3.13 Graphical programming

Topic	Description
Elements	If you select the face of a closed contour, you can insert a radius or chamfer at each corner of the contour.
Element information area	In the Element Information area, the control shows a rounding arc as **RND** contour element and a chamfer as **CHF** contour element.

1.3.14 CAD Viewer

Topic	Description
Unit of measure	Internally, **CAD Viewer** always uses mm for its calculations. If you select inches as the unit of measure, **CAD Viewer** will convert all values to inches.
Display	The **Show sidebar** icon enlarges the Sidebar window to half the size of the screen.
	The control always shows the **X**, **Y** and **Z** coordinates in the Element Information window. In 2D mode, the control grays out the Z coordinate.
Transferring machining positions	**CAD Viewer** also recognizes circles that consist of two semi-circles as machining positions.
Workpiece preset and workpiece datum	You can save the information on the workpiece preset and workpiece datum to a file or to the clipboard without having to resort to CAD Import (software option 42).

1.3.15 User aids

Topic	Description
The **Structure** column in the **Program** workspace	The structure contains the NC functions **APPR** and **DEP** as structure elements.
	The control shows comments in the structure inserted within structure elements.
	If you select structure elements in the **Structure** column, the control also marks the corresponding NC blocks in the NC program. Use the **CTRL+SPACE** key shortcut to stop marking. If you press **CTRL+SPACE** again, the control restores the marked selection.
The **Search** column in the **Program** workspace	The **Match whole words only** checkbox determines that the control shows only exact matches. If, for example, you search for **Z+10**, the control ignores **Z+100**.
	If in the **Search and replace** function you use **Find next**, the control highlights the first result in purple.
	If you do not enter a value for **Replace with:**, the control deletes the value searched for and to be replaced.
Program comparison	If you select several NC blocks during the program comparison, you can load all NC blocks simultaneously.
Keyboard shortcuts	The control provides additional keyboard shortcuts to mark NC blocks and files.
Context menu	When you open or save a file in a selection window, the control displays the context menu.
	You can perform file functions in the **Save as** window using the context menu.
Cutting data calculator	You can load the tool name from the cutting data calculator.
	If you press the enter key in the cutting data calculator, the control selects the next element.
Message menu	In the expanded notification menu, the control displays information about the NC program in a separate area outside of the **Details**.

1.3.16 Simulation workspace

Topic	Description
Workpiece position window	You can use a button to select a workpiece preset from the preset table.
	The control displays the input fields below each other instead of next to each other.
Finished part	The control can display a finished part in **Machine** mode of the **Simulation** workspace.
Depiction of tools	The control takes into account the following columns of the tool table for the simulation: ■ **R_TIP** ■ **LU** ■ **RN**
Dwell time	In the simulation of the **Editor** operating mode, the control takes dwell times into account. The control does not dwell during the program test, but adds the dwell times to the program run time.
NC functions	The NC functions **FUNCTION FILE** and **FN 27: TABWRITE** (ISO: **D27**) are effective in the **Simulation** workspace.
Simulation of turning cycles(option 50)	The control shows remaining residual material during turning cycles also with the machining operations **Q215=1** and **Q215=2**.

1.3.17 Touch probe functions in the Manual operating mode

Topic	Description
Aligning the rotary table	If you align the rotary table after a manual touch probe function, the control remembers the selected type of rotary axis positioning and the feed rate.
Applying values	If you correct the preset or datum after a manual touch probe function, the control shows a symbol behind the adopted value.

1.3.18 Programmable touch probe cycles

Topic	Description
Cycle **451MEASURE KINEMATICS** (ISO: **G451**, option 48)	If KinematicsComp (software option 52) is active, the log of Cycle **451 MEASURE KINEMATICS** (ISO: **G451**, option 48) shows the active compensations of the angular position errors (**locErrA**/**locErrB**/**locErrC**).
Cycle **451 MEASURE KINEMATICS** (ISO: **G451**) and Cycle **452 PRESET COMPENSATION** (ISO: **G452**, option 48)	The log of Cycles **451 MEASURE KINEMATICS** (ISO: **G451**) and **452 PRESET COMPENSATION** (ISO: **G452**, option 48) contains diagrams with the measured and optimized errors of the individual measuring positions.
Cycle **453 KINEMATICS GRID** (ISO: **G453**, option 48)	Cycle **453 KINEMATICS GRID** (ISO: **G453**, option 48) allows you to use the mode **Q406=0** even without KinematicsComp (software option 52).
Cycle **460 CALIBRATION OF TS ON A SPHERE** (ISO: **G460**)	Cycle **460 CALIBRATION OF TS ON A SPHERE** (ISO: **G460**) determines the radius and, if required, the length, the center offset and the spindle angle of an L-shaped stylus.
L-shaped stylus	Cycles **444 PROBING IN 3-D** (ISO: **G444**) and **14xx** support probing with an L-shaped stylus.

1.3.19 Pallet machining and job lists

Topic	Description
Batch Process Manager (option 154)	If you check the pallet table in Batch Process Manager (option 154) with Dynamic Collision Monitoring (DCM, option 40), the control takes the software limit switches into account.

1.3.20 Program run

Topic	Description
The **Open in the editor** button	The **Open in the editor** button in the **Program Run** operating mode opens the currently displayed NC program, including called NC programs.
Returning to the contour	In the machine parameter **restoreAxis** (no. 200305), the machine manufacturer defines in which sequence of axes the control approaches the contour again.

1.3.21 Tables

Topic	Description
Tables operating mode	The **M** and **S** statuses are highlighted in color only for the active application, and gray for the other applications.
	You can close all applications except for **Tool management**.
	The **Mark row** button has been added.
	In the **Presets** application, the **Lock record** toggle switch has been added.
Table workspace	You can change the column width using an icon.
	In the settings of the **Table** workspace you can enable or disable all table columns and restore the default format.
Form workspace	If a table column offers two input options, the control shows the options in the **Form** workspace as toggle switches.
TABDATA	You can use the **TABDATA** functions for read- and write-access to the preset table.

1.3.22 Settings application

Topic	Description
Code number	When you enter a code number in the **Settings** application, the control displays a load icon.
Network	You can export and import existing network configurations in the **Network settings** window.
Secure connections	In the **DNC** menu item of the **Settings** application, the **Secure connections for users** area has been added. These functions can be used to define settings for secure connections via SSH.
	In the **Certificate and keys** window you can select a file with additional public SSH keys in the **Externally administered SSH key file** area. This allows you to use SSH keys without needing to transmit them to the control.

1.3.23 Machine parameters

Topic	Description
Secure connections	The machine manufacturer uses the machine parameters **allowUnsecureLsv2** (no. 135401) and **allowUnsecureRpc** (no. 135402) to define whether the control disables non-secure LSV2 or RPC connections even if user administration is not active. These machine parameters are included in the data object **CfgDncAllowUnsecur** (135400).
Clear NC blocks	The optional machine parameter **warningAtDEL** (no. 105407) is used to define whether the control shows a confirmation request in a pop-up window when deleting an NC block.

2

Software 81762x-18

2.1 New functions

2.1.1 User's Manual as integrated product aid: TNCguide

Topic	Description
TNCguide	You can open **TNCguide** for the current context. Context-sensitive help means that the relevant information is displayed directly (e.g., for the selected item or the current NC function). Using the **Help** icon, you can select an item for which to display information. When you press the **HELP** key, the control will display information on the selected NC function.

2.1.2 Operation

Topic	Description
Hardware requirements	To install or update software version 18, a control with a hard disk size of at least 30 GB is required.
Announcement: **SIK2** plug-in board	Software version 18 SP1 introduces the **SIK2** plug-in board. For controls with **SIK2**, the software options are identified by new four-digit numbers. As long as both **SIK** and **SIK2** are available, both software option numbers will be indicated in the User's Manual, for example (#18 / #3-03-1).

2.1.3 Status displays

Topic	Description
The **Status** workspace	Using the **Configure the layout** icon in the **Status** workspace, you can add or remove columns and arrange the areas in columns.

2.1.4 Manual operation

Topic	Description
Unbalance functions (#50 / #4-03-1)	The control provides manual cycles that allow you to determine the unbalance in the current fixture. The control suggests the mass and position of the compensation weight.

Programming fundamentals

Topic	Description
The **Text editor** workspace	The **Text editor** workspace is available in the **Editor** operating mode. In the **Text editor** you can create and edit data of the following types: ■ Text files, such as *.txt ■ Format files, such as *.a
Settings in the **Program** workspace	You can deactivate the auto-complete function in Text editor mode.
	You can select whether the control is to display help graphics as pop-up windows or in the **Help** workspace only.
	You can select whether the control is to add an informational comment to an NC sequence, such as the name of the NC sequence.
	You can select whether the control will dim unavailable NC functions in the **Insert NC function** window or hide them (e.g., for software options that are not enabled).
	You can select whether the control will enclose path information in quotation marks by default for the following NC functions: ■ **CALL PGM** (ISO: **%**) ■ Cycle **12 PGM CALL** (ISO: **G39**) ■ **FN 16: F-PRINT** (ISO: **D16**) ■ **FN 26: TABOPEN** (ISO: **D26**)
	If a touchscreen is used, the control will display a context-sensitive virtual keyboard. A selection menu allows you to select the position of the virtual keyboard in the workspace or to hide the virtual keyboard.
Display of the NC program	In the machine parameter **lineBreak** (no. 105404), you define whether the control will display multi-line NC functions without or with line breaks.

2.1.5 Tools

Topic	Description
Tool type	The tool type **Side milling cutter** (**MILL_SIDE**) has been added.
Tool model (#140 / #5-03-2)	You can add 3D models for drilling or milling tools as well as workpiece touch probes. The control can display tool models in simulation and take them into account in calculations, for example when performing Dynamic Collision Monitoring (DCM (#40 / #5-03-1)).

2.1.6 Milling cycles

Topic	Description
Cycle **1274 OCM CIRCULAR SLOT** (ISO: **G1274**) (#167 / #1-02-1)	This cycle allows you to define a circular slot that is then used as a pocket or boundary for face milling in conjunction with other OCM cycles.

2.1.7 Coordinate transformation

Topic	Description
TRANS RESET	Use the NC function **TRANS RESET** to reset all simple coordinate transformations simultaneously.

2.1.8 Files

Topic	Description
The **Files** operating mode	With the settings of the **Files** operating mode, you can define whether the control will display hidden and dependent files, such as the tool-usage file ***.t.dep**.

2.1.9 Collision monitoring

Topic	Description
Combining fixtures	The **New Fixture** window allows combining several fixtures and saving them as a new fixture. This enables realizing and monitoring complex clamping situations.
FUNCTION DCM DIST (#140 / #5-03-2)	With the **FUNCTION DCM DIST** NC function, you can reduce the minimum distance between the tool and the fixture for Dynamic Collision Monitoring (DCM (#40 / #5-03-1)).

2.1.10 Programming with variables

Topic	Description

FN 18: SYSREAD (ISO: **D18**)	The **FN 18: SYSREAD** (ISO: **D18**) functions have been extended:
	■ **FN 18: SYSREAD (D18) ID10 NR10**: Counts the number of executions of the current program section
	■ **FN 18: SYSREAD (D18) ID245 NR1**: Current nominal position of an axis (**IDX**) in the REF system
	■ **FN 18: SYSREAD (D18) ID370 NR7**: Reaction of the control if a probing point is not reached during a programmable touch-probe cycle **14xx**
	■ **FN 18: SYSREAD (D18) ID610**: Values of various machine parameters for **M120**
	  ■ **NR53**: Radial jerk at normal feed rate
	  ■ **NR54**: Radial jerk at high feed rate
	■ **FN 18: SYSREAD (D18) ID630**: SIK information of the control
	  ■ **NR3**: SIK generation **SIK** or **SIK2**
	  ■ **NR4**: Specifies whether and how often a software option (**IDX**) has been enabled on controls with **SIK2**
	■ **FN 18: SYSREAD (D18) ID990 NR28**: Current tool spindle angle
	■ **FN 18: SYSREAD (D18) ID10950 NR6**: Selected file in the **TSHAPE** column of the tool table for the current tool (#140 / #5-03-2)

2.1.11 Graphical programming

Topic	Description
Importing contours into graphical programming	It is possible to import NC blocks that contain NC functions for coordinate transformation into the graphical programming environment.

2.1.12 ISO

Topic	Description
The **Insert NC function** window	The **Insert NC function** window allows you add ISO syntax, too. Using the NC function keys, you can insert the corresponding ISO syntax (e.g., by pressing the **L** key for **G01**).

2.1.13 User aids

Topic	Description
Context menu	The **Insert NC function** window features a context menu.

2.1.14 The Simulation workspace

Topic	Description
The **Simulation settings** window	The **Optimized saving of STL** (#152 / #1-04-1) toggle switch allows you to output a simplified STL file. These STL files have been adapted to the **BLK FORM FILE** function; for example, they contain a maximum of 20,000 triangles.

2.1.15 Touch probe functions in the Manual operating mode

Topic	Description
The **Change the preset** window	In the **Change the preset** window, you can discard the previous probing position and activate a new preset with the **Apply changes and delete existing probe objects** button.

2.1.16 Program run

Topic	Description
Retracting the tap	If the NC program stops during tapping, the control will display the **Tool Retract** button. When you select that button and press the **NC Start** key, the control will automatically retract the tool.

2.1.17 Tables

Topic	Description
The **Form** workspace	Using the **Configure the layout** icon in the **Form** workspace, you can add or remove columns and arrange the areas in columns.
Tool table	You can use the **TSHAPE** column of the tool table to select a 3D file as the tool model (#140 / #5-03-2). This allows the control to display complex tools in simulation and take them into account for Dynamic Collision Monitoring (DCM (#40 / #5-03-1)).
Freely definable tables	The **Edit table characteristics** icon allows you to, for example, insert new columns into freely definable tables.
Machine manufacturer settings	The machine manufacturer uses the machine parameter **CfgTableCell-Lock** (no. 135600) to define whether and in which cases individual table cells are locked or write-protected. On some machines, you cannot change the tool type once a tool has been inserted into the machine.
	Using the optional machine parameter **CfgTableCellCheck** (no. 141300), the machine manufacturer can define rules for table columns. This machine parameter allows to define columns as required fields or to reset them automatically to a default value. If a rule is violated, the control displays a note icon.

2.1.18 Override controller

Topic	Description
Override controller	With the OC 310 override controller hardware extension, the control allows the following: ■ Use the dial to manipulate the feed rate and/or rapid traverse ■ Start NC programs with the integrated **NC Start** button ■ Receive tactile responses through vibrations ■ Use breakpoints to define conditional stops ■ Resume the NC program by increasing the override

2.1.19 Integrated functional safety (FS)

Topic	Description
SLP safety function (safely limited position)	In machine parameter **safeAbsPosition** (no. 403130), the machine manufacturer defines whether the **SLP** safety function is activated for an axis. If the **SLP** safety function is inactive, the axis is monitored by functional safety (FS) without a check after startup. The axis is identified by means of a gray warning triangle.

2.1.20 HEROS operating system

Topic	Description
HEROS menu	In the HEROS settings, you can adjust the screen brightness of the control.
	In the **Screenshot settings** window, you can define under which path and file name the control saves screenshots. The file name can contain a placeholder (e.g., %N for sequential numbering).
	The HEROS tool **Diffuse** has been added. You can compare and merge text files. This tool is provided as an addition to the **program comparison** function for NC programs.

2.2 Modified or extended functions

2.2.1 Operation

Topic	Description
Dark Mode	In the machine parameter **darkModeEnable** (no. 135501), the machine manufacturer defines whether **Dark Mode** is available for selection.
Title bar of the workspaces	The control groups the icons of the title bar depending on the size of the workspace in a selection menu.

2.2.2 Status displays

Topic	Description
The **Positions** workspace	If the handwheel is active, the control shows a symbol next to the selected axis in the **Positions** workspace. The symbol indicates whether you can move the axis with the handwheel.
	When you move the axes while **M136** is active, the control will display the feed rate in mm/rev in the **Positions** workspace and on the **POS** tab of the **Status** workspace.
	When a pallet preset is active, the control displays an icon with the number of the active pallet preset in the **Positions** workspace.
Status overview on the TNC bar	You can select the position display mode in the status overview on the TNC bar independently of the **Positions** workspace (e.g., **Actual pos. (ACT)**).
The **Status** workspace	On the **FN 16** tab of the **Status** workspace, you can select the **Clear** button to clear the **Output** area.
	The **QPARA** tab can show 22 instead of 10 variables for each area.
	On the **MON** tab of the **Status** workspace, the histogram shows the entire signal range, using the colors of the relative display (#155 / #5-02-1).
	If the optional columns **WPL-DX-DIAM** and **WPL-DZL** of the turning-tool table exist, the control shows the values of these columns on the **Tool** tab of the **Status** workspace (#50 / #4-03-1).

2.2.3 Manual operation

Topic	Description
Handwheel	If you select **Manual** operating mode, the control deactivates the handwheel.

2.2.4 Programming fundamentals

Topic	Description
The **Editor** operating mode	You can change the tab order in the **Editor** operating mode.
The **Program** workspace	On the title bar of the **Program** workspace, the control shows icons for the **Cut**, **Copy** and **Paste** functions.
	While editing an NC block, you can undo individual changes made to syntax elements by selecting **Undo**.
The **Insert NC function** window	During searches, the control also displays search results in the **Insert NC function** window that contain the search term, and replacement functions as well as related or equivalent functions.
Help graphic	When you are editing an NC block, the control shows for some NC functions a help graphic in a pop-up window that illustrates the current syntax element. From this pop-up window, you can open the **Help** workspace or TNCguide.
Text editor mode	When you enter any character in Text editor mode, the control will insert a new line.
	When you program a cycle using the active auto-complete function, you can select the **Only downwardly-compatible cycle parameters** or **With optional cycle parameters** option. Optional cycle parameters can also be added later.
	In the selection menu of the Text editor mode, the control displays possible values in addition to the available syntax element (e.g., for the letter **M**).
	The control displays a help graphic in Text editor mode, too.
	In Text editor mode, you can insert line breaks.

2.2.5 Tools

Topic	Description
Tool data	The **thread-turning tool** turning tool type includes the parameter **SPB-Insert** (#50 / #4-03-1).
Indexed tools	In the **Insert tool** window, the **Index** checkbox was added. When you enable this checkbox, the control will add the next free index number.
	When you create an indexed tool, the control will copy the tool data from the previous table row. The previous table row may be the main tool or an existing indexed tool.
	If you delete a main tool, the control will delete all associated indexed tools as well.
Tool-usage test	The control displays the **Refresh** icon in the **Tool usage** and **Tool check** areas of the **Tool check** column. You can create a tool-usage file and run a tool-usage test.

2.2.6 Programming techniques

Topic	Description
NC sequences	You can activate or deactivate write protection for NC sequences.

2.2.7 Contour and point definitions

Topic	Description
SEL CONTOUR	You can also define subcontours as **LBL** subprograms within the complex **SEL CONTOUR** contour formula.
PATTERN DEF	The **Insert NC function** window shows every pattern definition of the **PATTERN DEF** function separately.
Cycle **220 POLAR PATTERN** (ISO: **G220**) and Cycle **221 CARTESIAN PATTERN** (ISO: **G221**)	The machine manufacturer can hide the cycles **220 POLAR PATTERN** (ISO: **G220**) and **221 CARTESIAN PATTERN** (ISO: **G221**). We recommend using the **PATTERN DEF** function.

2.2.8 Milling cycles

Topic	Description
Cycle **225 ENGRAVING** (ISO: **G225**)	The input value **1** has been added to parameter **Q515 FONT** in Cycle **225 ENGRAVING** (ISO: **G225**). Use this input value to select the **LiberationSans-Regular** font.
Cycle **208 BORE MILLING** (ISO: **G208**) and Cycles **127x** OCM standard figure cycles (#167 / #1-02-1)	You can enter symmetric tolerances for nominal dimensions, such as **10+-0.5**.
Cycle **287 GEAR SKIVING** (ISO: **G287**) (#157 / #4-05-1)	Cycle **287 GEAR SKIVING** (ISO: **G287**) (#157 / #4-05-1) has been extended: ■ When you program the optional parameter **Q466 OVERRUN PATH**, the control will optimize the approach and idle travel paths automatically. This will reduce machining times. ■ Two columns have been added to the prototype of the technology table: ■ **dK**: Angular offset of the workpiece in order to machine one side of the tooth flank only. This can be used to increase the surface quality. ■ **PGM**: Profile program for a custom tooth flank line, for example to realize crowning of the tooth flank. ■ After each step, the control displays the number of the current cut and the number of remaining cuts in a pop-up window.
Cycle **286 GEAR HOBBING** (ISO: **G286**) (#157 / #4-05-1) and Cycle **287 GEAR SKIVING** (ISO: **G287**) (#157 / #4-05-1)	The machine manufacturer can configure a deviating automatic **LIFTOFF** for Cycles **286 GEAR HOBBING** (ISO: **G286**) (#157 / #4-05-1) and **287 GEAR SKIVING** (ISO: **G287**) (#157 / #4-05-1).

2.2.9 Mill-turning cycles (#50 / #4-03-1)

Topic	Description
Cycle **800 ADJUST XZ SYSTEM** (ISO: **G800**) (#50 / #4-03-1)	Cycle **800 ADJUST XZ SYSTEM** (ISO: **G800**) (#50 / #4-03-1) has been extended: ■ The input range of the parameter **Q497 PRECESSION ANGLE** has been extended from four to five decimal places. ■ The input range of the parameter **Q531 ANGLE OF INCIDENCE** has been extended from three to five decimal places.

2.2.10 Files

Topic	Description
File functions	If file functions are available for a selected folder or file, the control will display three dots below the icon.
	If you copy a file and then paste it to the same folder, the control adds the suffix **_1** to the file name. The control increments the number sequentially for each consecutive copy.
File preview	The control indicates by means of symbols in the file preview whether the entire file or only a part of it is displayed.
The **Document** workspace	The **Document** workspace includes a file information bar where the file path is shown.
	For PDF files, additional functions, such as searching or scaling, are available in the **Document** workspace.
	In the **Internet** window, you can mark URLs as bookmarks.
Quick selection workspaces	The **Quick selection** workspace in the **Editor** operating mode is subdivided into the following areas: ■ **NC programs** ■ **New graphical programming** ■ **New text file** ■ **Jobs**
	The **Create new table** function of the **Quick selection new table** workspace was revised. Now, you can, for example, search for table types and add favorites.

2.2.11 Monitoring

Topic	Description
Component monitoring (#155 / #5-02-1)	If a component has not been configured or cannot be monitored, the control displays the corresponding machining operation in gray in the heatmap.
Process monitoring	The predefined HEIDENHAIN monitoring tasks have been updated and extended, for example by signals and processes. The machine manufacturer can configure additional monitoring tasks.
	It is no longer necessary to select reference machining explicitly. You can classify recordings as good or bad parts. The control will automatically use the first ten "good" recordings as reference machining.
	Recordings of machining operations can be exported manually or automatically to a log file.
	Recordings and settings of prior software versions are not compatible with software version 18.

2.2.12 Miscellaneous functions

Topic	Description
Miscellaneous functions for the spindle	In turning mode, miscellaneous functions for the turning spindle must be programmed using different numbers (e.g., **M303** instead of **M3** (#50 / #4-03-1)). The machine manufacturer defines the numbers to be used. Using the optional machine parameter **CfgSpindleDisplay** (no. 139700), the machine manufacturer defines the miscellaneous function numbers to be displayed in the status display.
The **Manual operation** application	The machine manufacturer uses the optional machine parameter **forbidManual** (no. 103917) to define which miscellaneous functions are allowed in the **Manual operation** application and are available in the selection menu.

2.2.13 Programming with variables

Topic	Description
Formulas	If you press the spacebar while using the **Formula**, **String formula** and **Contour formula** NC functions, the control displays all currently usable syntax elements in the action bar.
	Press the **-/+** key to change the algebraic sign in formulas.

2.2.14 Graphical programming

Topic	Description
The **Contour settings** window	The control will save the settings made in the **Contour settings** window permanently. Only the **Plane** and **Diameter programming** settings are not saved.

2.2.15 CAD Viewer

Topic	Description
CAD Import (#42 / #1-03-1)	When you select contours and positions in **CAD Viewer**, you can rotate the workpiece using touch gestures. While you are using touch gestures, the control will not display any element information.
	CAD Import (#42 / #1-03-1) subdivides contours that are not located in the working plane into individual sections. **CAD Viewer** creates straight lines **L** and circular arcs that are as long as possible. The resulting NC programs are often much shorter and clearer than NC programs generated by CAM. Thus, the contours are better suited for cycles, such as the OCM cycles (#167 / #1-02-1).
	CAD Import outputs the radii of the circular arcs as comments. At the end of the generated NC blocks, CAD Import displays the smallest radius to help you select the most suitable tool.
	In the **Find circle centers by diameter range** window, you can filter the data by position depth values.

2.2.16 ISO

Topic	Description
ISO programming	In connection with ISO programming, the control provides the following functions: ■ Auto-complete ■ Color highlighting of syntax elements ■ Structure

2.2.17 User aids

Topic	Description
Comments and structuring items	You can insert line breaks within comments or structuring items.
The **Structure** column	You can use the context menu to mark structuring items in the **Structure** column. The control will also mark all corresponding NC blocks.
Search column in the **Program** workspace	If you use **Search and replace** while NC programs are open, the control will close them.
	The limit of the **Replace all** function was extended from 10,000 to 100,000.
Calculator	You can use the calculator to convert mm values to inch values and vice versa.
	The calculator features separate buttons for the arcsin, arccos and arctan trigonometric functions.
Message menu	In the message menu, you can use the **Setting for autosave** button to specify up to five error numbers. The control will automatically create a service file if one of these errors occurs.
	Using a toggle switch, you can define whether the control will save data from process monitoring (#168 / #5-01-1) for the current NC program in the service file.

2.2.18 The Simulation workspace

Topic	Description
The **Simulation settings** window	In the **Editor** operating mode, the **Simulation** workspace can be open for only one NC program at a time. With this NC program the control displays the **Control-in-operation** icon next to the program name. If you want to open the workspace on a different tab, the control prompts you for confirmation. The query depends on the simulation settings and the status of the active simulation.
Preset	Before acknowledging a power interruption, you can select a preset for the **Simulation** workspace.
Advanced checks	Within the **Advanced checks** function, you can activate the following checks individually: ■ Material removal at rapid traverse ■ Collisions between the tool carrier or tool shank and the workpiece ■ Collisions between the tool and the fixture

2.2.19 Touch probe functions in the Manual operating mode

Topic	Description
Probe process	When you select a manual touch-probe function, the control automatically suggests the probing direction last used for this function.
	After probing, the control will always display the axis probed in the **Measuring** area.
	If a probing point could not be reached, you can continue probing by pressing the **NC Start** key.
Automatic probing method	When you select automatic probing within a touch-probe function, the control will use the sum of the value in the **SET_UP** column and the stylus tip radius as the set-up clearance. The set-up clearance cannot be less than the value in the **SET_UP** column of the touch-probe table.
Plane over cylinder (PLC) touch-probe function	For the **Plane over cylinder (PLC)** touch-probe function, the second measurement is by default in the inverse direction of the first measurement. Thus, pre-positioning in the probing plane is not necessary because the control will use the current angle as the start angle.
Calibrating the touch probe	If you have used a calibration sphere to calibrate the radius of a touch probe, the control will automatically select the 3D Calibration function (#92 / #2-02-1).
The **Change the preset** window	In the **Change the preset** window, you can enter a different preset.

2.2.20 Touch-probe cycles for workpieces

Topic	Description
Touch-probe cycles **14xx** for determining a workpiece misalignment and for acquiring the preset	You can enter symmetric tolerances for nominal dimensions, such as **10+-0.5**.
Cycle **441 FAST PROBING** (ISO: **G441**)	Cycle **441 FAST PROBING** (ISO: **G441**) now features the parameter **Q371 TOUCH POINT REACTION**. This parameter defines the reaction of the control in cases where the stylus is not deflected.
	Using the parameter **Q400 INTERRUPTION** in Cycle **441 FAST PROBING** (ISO: **G441**), you can define whether the control will interrupt program run and display a measuring log. The parameter is effective in conjunction with the following cycles: ■ Cycle **444 PROBING IN 3-D** (ISO: **G444**) ■ Touch-probe cycles **45x** for kinematics measuring ■ Touch-probe cycles **46x** for calibrating the workpiece touch probe ■ Touch-probe cycles **14xx** for determining a workpiece misalignment and for acquiring the preset

2.2.21 Touch-probe cycles for tools

Topic	Description
Tool measurement cycles **48x**	Using the optional machine parameter **maxToolLengthTT** (no. 122607), the machine manufacturer defines a maximum tool length for tool touch probe cycles. If a tool has been defined in the tool table with a length of **L = 0**, the control will use the value of the machine parameter as the starting point for a rough length measurement. Then, a fine measurement will be performed.
	Using the optional machine parameter **calPosType** (no. 122606), the machine manufacturer defines whether the position of parallel axes and changes in the kinematics should be considered for calibration and measuring. A change in kinematics might for example be a head change.

2.2.22 Touch-probe cycles for measuring the kinematics

Topic	Description
Cycle **451 MEASURE KINEMATICS** (ISO: **G451**) (#48 / #2-01-1) and **452 PRESET COMPENSATION** (ISO: **452**) (#48 / #2-01-1)	Cycles **451 MEASURE KINEMATICS** (ISO: **G451**) (#48 / #2-01-1) and **452 PRESET COMPENSATION** (ISO: **452**) (#48 / #2-01-1) save the measured position errors of the rotary axes in the QS parameters **QS144** to **QS146**.

2.2.23 Program run

Topic	Description
Feed-rate limitation	The button for feed-rate limitation and the associated functions (previously **FMAX**) were renamed to **F LIMIT**.
Execution cursor	The execution cursor is always displayed in the foreground. The execution cursor may cover or hide other icons.
Presets	When running an NC program in **Single Block** mode, you can edit the preset table. Before editing, the control displays a prompt where you must confirm that you want to abort program run.

2.2.24 Tables

Topic	Description
Creating a new table	When you create a new table in the file manager, the table does not contain information on the required columns yet. When you open the table for the first time, the **Incomplete table layout** window will open in the **Tables** operating mode. In the **Incomplete table layout** window, a selection menu allows you to select a table template. The control shows which table columns are added or removed, if applicable.
Editing a table	To edit the contents of a table, you can also double-tap or double-click the table cell. The control displays the **Editing disabled. Enable?** window. You can enable the values for editing or abort the process.
	If you copy or cut a table row in the **Tables** operating mode, the control provides the **Overwrite** or **Append** function for pasting.
	If you select the contents of a cell in a selection window, the control displays the **Delete entry** button.
The **Table** workspace	The **Change column width** function remains active if you select a different column.
The **Form** workspace	In the **Form** workspace for tables, the control displays help graphics that show the effect of the selected grinding tool parameters.
Accessing table values	In the **TABDATA WRITE**, **TABDATA ADD** and **FN 27: TABWRITE** (ISO: **D27**) NC functions, you can enter values directly.
Tool management	You cannot delete any tools that have been entered into the pocket table. The button is dimmed.
	The selection window for 3D files includes a search function.
	If you insert a new table row in tool management using the **Insert tool** button, the control will suggest the next free row number.
	The control displays icons for the **TO** orientations of the dressing tools (#156 / #4-04-1).
	In some operating modes and applications, you can use the **Tools** button to switch to **Tool management**.

2.2.25 The Settings application

Topic	Description
OPC UA NC Server (#56-61 / #3-02-1*)	Within the **OPC UA** menu item, a button is available to manually start or restart the **OPC UA NC Server**.
	The **OPC UA NC Server** allows you to create service files.
	You can validate 3D models for tools or tool carriers (#140 / #5-03-2).
	The **OPC UA NC Server** supports the **Aes128Sha256RsaOaep** and **Aes256Sha256RsaPss** security policies.
PKI Admin	If an attempt to connect to the **OPC UA NC Server** (#56-61 / #3-02-1*) fails, the control will store the client certificate on the **Rejected** tab. You can transfer the certificate directly to the **Trusted** tab without the need to transfer the certificates manually to the control.
	You can open **PKI Admin** from the **OPC UA** menu item.
	PKI Admin now includes the **Advanced settings** tab. You can define whether the server certificate should contain static IP addresses and allow connections without an associated CRL file.
Secure connections	The control uses an icon to indicate whether a connection configuration is secure or non-secure.
	In future software versions, the control will no longer support LSV2 protocols.
Configuration of the control's user interface	The following buttons have been added to the **Configurations** menu item: ■ **Save current settings** ■ **Restore last configuration**

2.2.26 User administration

Topic	Description
Login as a function user	Your IT administrator can set up a function user to facilitate connectivity to the Windows domain.
Connecting to a Windows domain	If you have connected the control to the Windows domain, you can export the required configurations for other controls.

2.2.27 Machine parameters

Topic	Description
Display of the machine parameters	In the **List** workspace, you can toggle between a structure and a table view of the configuration editor.
StretchFilter	Machine parameter **CfgStretchFilter** (no. 201100) has been removed.

3

Software 81762x-19

3.1 Modified or enhanced software options

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

Topic	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.

3.1.2 Interpolating spindle (#96 / #7-04-1)

Topic	Description
Adv. Spindle Interpol. software option (#96 / #7-04-1)	The **FUNCTION SHAPING** contour planing function has been added.
	FUNCTION TURNDATA CORR is also available with the Adv. Spindle Interpol. software option (#96 / #7-04-1).
	The turning-tool table is also available with the Adv. Spindle Interpol. software option (#96 / #7-04-1).

3.2 New functions

3.2.1 About the product

Topic	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 otherwise display a warning.

3.2.2 Status displays

Topic	Description
The **Status** workspace	In the **PROCMON** tab of the **Status** workspace, the control shows information on process monitoring (#168 / #5-01-1) in the **Program Run** operating mode. When process monitoring for the NC program is set up completely, you will get a compact overview of the current machining process.
	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.

3.2.3 Programming fundamentals

Topic	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.

3.2.4 Technology-specific NC programming (#156 / #4-04-1)

Topic	Description
Cylindrical grinding with **FUNCTION MODE GRIND**	Use **FUNCTION MODE GRIND** (#156 / #4-04-1) to activate cylindrical grinding and select a kinematic model. Cylindrical grinding on a milling machine enables you to perform complete machining tasks on a machine without rechucking. Cylindrical grinding helps you to attain higher accuracies and better surface definitions than with turning.

3.2.5 Path functions

Topic	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.

3.2.6 Grinding cycles (#156 / #4-04-1)

Topic	Description
Cycle **1011 DRESSING SIDE A/I** (ISO: **G1011**) (#156 / #4-04-1)	This cycle dresses the front face or shaft face of a grinding wheel. You define the dressing operation and the number of cycle calls after which dressing is performed. You can use this cycle only in dressing mode (**FUNCTION MODE DRESS**).
Cycle **1012 DRESSING D AND A/I** (ISO: **G1012**) (#156 / #4-04-1)	This cycle dresses the front face or shaft face and the diameter of a grinding wheel. You define the dressing operation and the number of cycle calls after which dressing is performed. You can use this cycle only in dressing mode (**FUNCTION MODE DRESS**).
Cycle **1041 LONG STROKE DEF.** (ISO: **G1041**) (#156 / #4-04-1)	This cycle defines the starting point and the reciprocating movement along a contour. The contour to be machined must be longer than the cutting edge of the grinding tool used. In combination with Cycle **1051 STEP. CYLIND. GRIND**, you can machine contours on the diameter, shoulder or plane surfaces.
Cycle **1042 SHORT STROKE DEF.** (ISO: **G1042**) (#156 / #4-04-1)	This cycle defines the starting point and the reciprocating movement along a cylindrical surface. The contour to be machined must be shorter or only a little longer than the cutting edge of the grinding tool used. In combination with Cycle **1053 CONTINOUS CYLIND. GRIND.**, you can machine contours on the diameter, shoulder or plane surfaces.
Cycle **1051 STEP. CYLIND. GRIND** (ISO: **G1051**) (#156 / #4-04-1)	This cycle defines the infeed movement of a cylindrical grinding operation and starts machining. Machining includes linear reciprocating movements and infeed movements. Cycle **1051 STEP. CYLIND. GRIND** performs the infeed incrementally at the reversal points of the reciprocating movement.
Cycle **1053 CONTINOUS CYLIND. GRIND.** (ISO: **G1053**) (#156 / #4-04-1)	This cycle defines the infeed movement of a cylindrical grinding operation and starts machining. Machining includes reciprocating movements and continuous infeed steps. This means that the infeed is even and performed without interruptions during the reciprocation movements.
Cycle **1040 END CYLIND. GRINDING** (ISO: **G1040**) (#156 / #4-04-1)	This cycle resets the following settings that you have defined in the cylindrical grinding cycles: ■ Reciprocating and infeed movements ■ Precession angle ■ Encoders and acoustic emission sensors Use this cycle to return an inclined axis to the initial position and to automatically retract it to the safety position.

3.2.7 Programming techniques

Topic	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 clearly 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.

3.2.8 Files

Topic	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.

3.2.9 Text editor

Topic	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.

3.2.10 Collision monitoring

Topic	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.

3.2.11 Monitoring (#168 / #5-01-1)

Topic	Description
Process monitoring (#168 / #5-01-1)	The control provides the **Filter** column, in which you can filter the monitored operations by date or status, for example.
	The control provides the **Form** column with settings or the monitoring tasks. The contents and options depend on the active table mode.
	The table in the **Process Monitoring** workspace has two modes: **Setup table** and **Runtime table**. You can switch between the modes with icons. In **Setup table** mode, you can see all monitoring sections of the NC program and the defined monitoring tasks. In **Runtime table** mode, you see the machining processes being monitored and the corresponding information.
	Process monitoring includes the monitoring task **Feed per tooth — Display**.
	The **MinMaxTolerance**, **Standard deviation** and **Absolute deviation** processes have been combined in one **Tunnel** process. The previous processes continue to exist as parameterization options.

3.2.12 Multiple-axis machining

Topic	Description
Contour planing with **FUNCTION SHAPING** (#96 / #7-04-1)	Contour planing, also known as shaping, enables you to create sealing surfaces with a high surface definition, for example. When **FUNCTION SHAPING** is active, the control automatically moves the tool toward the contour during traverse movements. Using **FUNCTION SHAPING**, this automatic tracking also enables you to perform engraving, engine turning, or beveling.
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.

3.2.13 Programming with variables

Topic	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.

FN 18: SYSREAD (ISO: **D18**)	The **FN 18: SYSREAD** (ISO: **D18**) functions have been enhanced:
	■ **FN 18: SYSREAD (D18) ID71 NR20**: machining information for dressing (#156 / #4-04-1)
	  ■ **IDX17**: wheel side to be dressed
	  ■ **IDX18**: grinding tool type
	  ■ **IDX19**: number of the active dressing cycle
	■ **FN 18: SYSREAD (D18) ID720 NR0 IDX1**: status of a reciprocating movement during grinding (#156 / #4-04-1)

3.2.14 The Contour graphics workspace

Topic	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.

3.2.15 Opening CAD files with CAD Viewer

Topic	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

3.2.16 ISO

Topic	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.

3.2.17 User aids

Topic	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.

3.2.18 The Simulation workspace

Topic	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.

3.2.19 Touch probes

Topic	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**.

3.2.20 Touch probe functions in the Manual operating mode

Topic	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.

3.2.21 Touch-probe cycles for workpieces

Topic	Description
Cycle **1403 RECTANGLE PROBING**	This cycle determines the center, width and length of a rectangle. The control probes two opposing touch points in each case.

3.2.22 Program run

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

3.2.23 Tables

Topic	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.

3.2.24 Electronic handwheel

Topic	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**.

3.2.25 Integrated functional safety (FS)

Topic	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.

3.2.26 The Settings application

Topic	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.

3.2.27 Machine parameters

Topic	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 combination **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).
Process monitoring (#168 / #5-01-1)	With the optional machine parameter **autoExportType** (no. 141602), you can define a file type that the control will automatically use to export recordings.
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.

3.2.28 User administration

Topic	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 external systems (e.g., robots). Refer to your machine manual.

3.2.29 HEROS operating system

Topic	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 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.

3.2.30 Accessories

Topic	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.

3.3 Modified or extended functions

3.3.1 Status displays

Topic	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 **m** and **s** or **h** and **m**.

3.3.2 Programming fundamentals

Topic	Description
Marking an area using identical 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.

3.3.3 Technology-specific NC programming (#156 / #4-04-1)

Topic	Description
Dressing of grinding tools	You can dress grinding tools with assigned tool carrier kinematics.

3.3.4 Tools

Topic	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.

3.3.5 Contour and point definitions

Topic	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 automatically generated a file with the extension ***.hp.pnt.dep**. You can also use this file with software version 19.

3.3.6 Cycles for milling and drilling

Topic	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 slightly 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 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 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 chamfering 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.

Topic	Description
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.

3.3.7 Grinding cycles (#156 / #4-04-1)

Topic	Description
Cycle **1000 DEFINE RECIP. STROKE** (ISO: **G1000**) (#156 / #4-04-1)	The following parameters have been added to Cycle **1000 DEFINE RECIP. STROKE**: ■ **Q1003 RECIPROCATING STROKE**: The parameter defines the coordinate system in which the reciprocating stroke will be effective. You can choose between the **I-CS** input coordinate system or the **T-CS** tool coordinate system. ■ **Q1060 X COMPONENT**: X component of the direction vector for defining the reciprocating stroke ■ **Q1061 Y COMPONENT**: Y component of the direction vector for defining the reciprocating stroke ■ **Q1062 Z COMPONENT**: Z component of the direction vector for defining the reciprocating stroke
Cycle **1010 DRESSING DIAMETER** (ISO: **G1010**) (#156 / #4-04-1) and Cycle **1016 DRESSING OF CUP WHEEL** (ISO: **G1016**) (#156 / #4-04-1)	The parameter **Q253 F PRE-POSITIONING** has been added to Cycles **1010 DRESSING DIAMETER** and **1016 DRESSING OF CUP WHEEL**. This parameter allows you to define the traversing speed of the tool during approach, retraction and return movements.
Cycle **1015 PROFILE DRESSING** (ISO: **G1015**) (#156 / #4-04-1)	The following parameters have been added to Cycle **1015 PROFILE DRESSING**: ■ **Q1006 GRINDING WHEEL FACE**: This parameter allows you to select whether the control dresses the front face or shaft face. ■ **Q253 F PRE-POSITIONING**: This parameter allows you to define the traversing speed of the tool during approach, retraction and return movements.
Cycle **1017 DRESSING WITH DRESSING ROLL** (ISO: **G1017**) (#156 / #4-04-1)	The parameter **Q1028 OVERLAP** has been added to Cycle **1017 DRESSING WITH DRESSING ROLL**. If the width of the dressing roll is larger than the width of the grinding wheel, you can program an overlap. Thus, the control will use the entire width of the dressing roll.

3.3.8 Programming techniques

Topic	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.

3.3.9 Compensations

Topic	Description
FUNCTION TURNDATA CORR	With **FUNCTION TURNDATA CORR-WPL**, you can define a delta value in the X direction as a diameter value using the **DXL-DIAM:** syntax element.

3.3.10 Files

Topic	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 information	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.

3.3.11 Text editor

Topic	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.

3.3.12 Collision monitoring

Topic	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 attributes (e.g., colors).
Advanced checks	The **Workpiece collision** check has been renamed to **Collision between workpiece and tool**.

3.3.13 Monitoring (#168 / #5-01-1)

Topic	Description
Process monitoring (#168 / #5-01-1)	By default, the control displays six monitoring tasks.
	Process monitoring shows suggestion type notes (e.g., **Consider deleting all records for NC program**).
	The **Signal display** icon allows you to switch between the signal curve and the resulting value during program run.
	The selection options for saving the process data have been enhanced. If you select **Interval: Record each nth operation and critical operations**, the control will only record the process data of, for example, every fourth machining process and of processes affected by interruptions.
	If process monitoring is active and you interrupt the program run within a monitoring section, the control will deactivate process monitoring only for this monitoring section. For the following monitoring section, process monitoring is active again.

3.3.14 Programming with variables

Topic	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.

3.3.15 User aids

Topic	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.

3.3.16 Touch probe functions in the Manual operating mode

Topic	Description
Working plane is inconsistent! (#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.

3.3.17 Touch-probe cycles for workpieces

Topic	Description
Touch probe cycles **42x** and **43x**	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 **485 MEASURE LATHE TOOL**	Cycle **485 MEASURE LATHE TOOL** is also available without the software options Turning (#50 / #4-03-1) or Turning v2 (#158 / #4-03-2).
Cycle **1404 PROBE SLOT/ RIDGE** (ISO: G1404)	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.

3.3.18 Pallet machining and job lists

Topic	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.

3.3.19 Program run

Topic	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 interruption.
	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.

3.3.20 Tables

Topic	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 selected 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.

3.3.21 Electronic handwheel

Topic	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). The handwheel also shows the speed of the rotary table (#50 / #4-03-1).
Wireless handwheel	The control displays a warning if you connect a wireless handwheel with an already selected radio channel.

3.3.22 Integrated functional safety (FS)

Topic	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 configure the activation and deactivation.
Check axis positions	In the **Referencing** workspace, you can switch as desired between the **Referencing** and **Check axis positions** modes.

3.3.23 The Settings application

Topic	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 connection status in the information bar.
TNCscope	The **TNCscope** application can only be opened with machine manufacturer rights.

3.3.24 Machine parameters

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

3.3.25 User administration

Topic	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.

3.3.26 HEROS operating system

Topic	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 manufacturer rights.

HEIDENHAIN

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

Technical support FAX +49 8669 32-1000
Measuring systems ☏ +49 8669 31-3104
service.ms-support@heidenhain.de
NC support ☏ +49 8669 31-3101
service.nc-support@heidenhain.de
NC programming ☏ +49 8669 31-3103
service.nc-pgm@heidenhain.de
PLC programming ☏ +49 8669 31-3102
service.plc@heidenhain.de
APP programming ☏ +49 8669 31-3106
service.app@heidenhain.de

www.heidenhain.com

1373081 · 02 · A · 02 · 09/2024 · PDF