



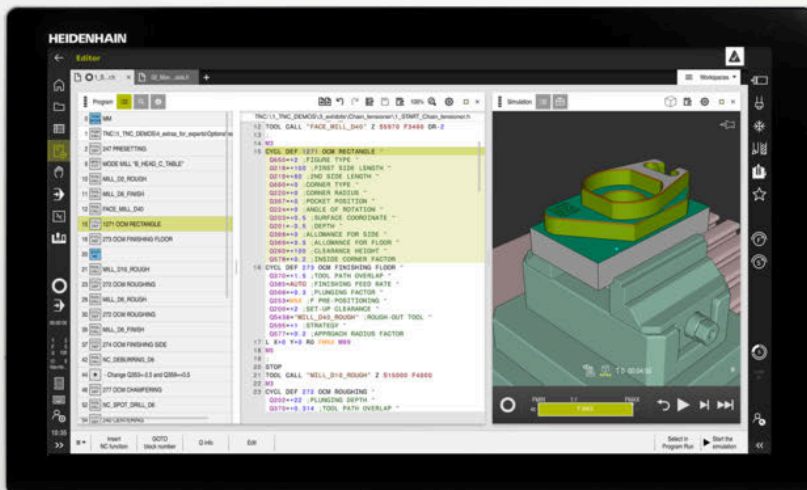
HEIDENHAIN



TNC7

Overview of New and Modified Software Functions

NC Software
81762x-17



English (en)
10/2022

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

Contents

1	Software 81762x-17.....	7
----------	--------------------------------	----------

1	Software 81762x-17	7
1.1	New software options	8
1.1.1	Model Aided Setup (option 159)	8
1.2	New functions	8
1.2.1	Programming fundamentals	8
1.2.2	Tools	9
1.2.3	Path Functions	9
1.2.4	Programming Techniques	9
1.2.5	Files	9
1.2.6	Variable Programming	10
1.2.7	ISO	10
1.2.8	User Aids	10
1.2.9	Simulation Workspace	10
1.2.10	Programmable Touch Probe Cycles	11
1.2.11	Program Run	11
1.2.12	Tables	11
1.2.13	Settings Application	12
1.2.14	Machine parameters	12
1.3	Modified or extended functions	12
1.3.1	Operation	12
1.3.2	Accessories	12
1.3.3	Status Displays	13
1.3.4	Powering On and Off	13
1.3.5	Programming fundamentals	14
1.3.6	Tools	14
1.3.7	Path Functions	15
1.3.8	Machining Cycles	15
1.3.9	Coordinate Transformation	15
1.3.10	Compensations	15
1.3.11	Files	16
1.3.12	Variable Programming	16
1.3.13	Graphical Programming	16
1.3.14	CAD-Viewer	17
1.3.15	User Aids	17
1.3.16	Simulation Workspace	18
1.3.17	Touch Probe Functions in the Manual Operating Mode	18
1.3.18	Programmable Touch Probe Cycles	18
1.3.19	Pallet Machining and Job Lists	19
1.3.20	Program Run	19
1.3.21	Tables	19
1.3.22	Settings Application	19
1.3.23	Machine parameters	20

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	<p>This software option is used to determine the position and misalignment of a workpiece with only one touch-probe function. You can probe complex workpieces with, for example, free-form surfaces or undercuts, which is not possible with all of the other touch-probe functions.</p> <p>The software option includes the Set up the workpiece touch probe function.</p>

1.2 New functions

1.2.1 Programming fundamentals

Topic	Description
Text editor	<p>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.</p> <p>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.</p>
Depiction of the NC program	<p>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.</p>
The Program settings window	<p>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.</p> <p>If the toggle switches in the Klartext area are active, the control skips the syntax elements Comment, Tool index and Linear superimposition.</p>

1.2.2 Tools

Topic	Description
Tool types	The following tool types have been added: <ul style="list-style-type: none"> ■ 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 a workshop). 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): <ul style="list-style-type: none"> ■ 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 Variable Programming

Topic	Description
FN 18: SYSREAD (ISO: D18)	<p>The FN 18: SYSREAD (ISO: D18) functions have been enhanced:</p> <ul style="list-style-type: none"> ■ 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 <ul style="list-style-type: none"> ■ 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 show the worktable and, if necessary, the fixtures, in Workpiece mode and with the Clamping situation toggle switch.

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 individually adapted. You can save and activate individual 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	<p>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.</p> <p>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.</p> <p>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.</p>
Feed rate limitation	<p>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.</p> <p>If the feed rate is limited using the FMAX button, the control displays MAX in square brackets.</p> <p>If the feed rate is limited using the F limited button, the control displays the active safety function in square brackets.</p>
Status workspace	<p>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.</p> <p>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.</p>
Handwheel	<p>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.</p>

1.3.4 Powering On and Off

Topic	Description
Shut down	<p>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.</p>

1.3.5 Programming fundamentals

Topic	Description
Input	<p>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.</p> <p>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.</p> <p>If you edit a value and press the backspace key, the control deletes only the last character and not the complete input.</p>
The Insert NC function window	<p>If software options are not enabled, the control shows unavailable contents in the Insert NC function window grayed out.</p> <p>In the areas Search result, Favorites and Last functions, the control shows the path of the NC functions.</p> <p>If you select an NC function and swipe to the right, the control displays the following file functions:</p> <ul style="list-style-type: none"> ■ Add to or remove from favorites ■ Open containing folder <p>Only when you search for an NC function</p>
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	<p>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.</p> <p>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.</p> <p>You can import tool tables of the TNC 640 as CSV files.</p>
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 : <ul style="list-style-type: none"> ■ 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	<p>The control shows the occupied memory and total memory of the drives in the navigation bar of the file management.</p> <p>The control shows STEP files in the preview area.</p> <p>When you cut a file or folder in the file management, the control grays out the icon of the file or folder.</p> <p>When you add a favorite or lock a file in the file management, the control displays an icon next to the file or folder.</p>
Quick selection workspace	<p>Tables for execution and simulation can be opened in the Quick selection workspace in the Tables operating mode.</p> <p>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.</p>

1.3.12 Variable Programming

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 mark structuring items in the Structure column, the control propagates the marking to 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
The 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 the Machine mode of the Simulation workspace.
Depiction of tools	The control takes into account the following columns of the tool table for the simulation: <ul style="list-style-type: none"> ■ R_TIP ■ LU ■ RN
Dwell time	In the Simulation function 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 active in the Simulation workspace.

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 451 MEASURE 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	<p>The M and S statuses are highlighted in color only for the active application, and gray for the other applications.</p> <p>You can close all applications except for Tool management.</p> <p>The Mark row button has been added.</p> <p>In the Presets application, the Lock record toggle switch has been added.</p>
Table workspace	<p>You can change the column width using an icon.</p> <p>In the settings of the Table workspace you can enable or disable all table columns and restore the default format.</p>
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	<p>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.</p> <p>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.</p>

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.

HEIDENHAIN

DR. JOHANNES HEIDENHAIN GmbH

Dr.-Johannes-Heidenhain-Straße 5

83301 Traunreut, Germany

☎ +49 8669 31-0

☎ +49 8669 32-5061

info@heidenhain.de

Technical support ☎ +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

