



HEIDENHAIN



TNC 620

Overview of New and Modified Software Functions

NC Software

73498x-02

81760x-01 to 81760x-08

English (en)
10/2020

About this document

This document describes the new and modified software functions of the TNC 620. Each software version is covered in a separate chapter.

The chapters are structured as follows:

- New software options
- New functions
- Modified functions

Each subsection begins with a reference to the User's Manuals for the respective software version.

The contents are sequenced as follows: The functions for programming and testing the NC programs are described first. These are followed by the contents covering program run and setup, as well as tables and tools. Finally, the settings options for the control are explained.

The contents are divided into categories based on the headlines in the User's Manuals. 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.

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 73498x-02.....	15
2	Software 81760x-01.....	23
3	Software 81760x-02.....	31
4	Software 81760x-03.....	41
5	Software 81760x-04.....	49
6	Software 81760x-05.....	59
7	Software 81760x-06.....	73
8	Software 81760x-07.....	83
9	Software 81760x-08.....	93

1	Software 73498x-02.....	15
1.1	New software options.....	16
	DXF Converter (option 42).....	16
	Remote Desktop Manager (option 133).....	16
	Cross Talk Compensation (CTC) (option 141).....	16
	Position Adaptive Control (PAC) (option 142).....	16
	Load Adaptive Control (LAC) (option 143).....	16
	Motion Adaptive Control (MAC) (option 144).....	16
	Active Chatter Control (ACC) (option 145).....	16
1.2	New functions.....	17
	Machining cycles.....	17
	Touch probe cycles (option 17).....	17
	Miscellaneous functions.....	17
	Program functions.....	18
	Programming Q parameters.....	18
	File management.....	18
	Moving the machine axes.....	18
	Manual touch probe cycles (option 17).....	18
	Accessories.....	19
	MOD functions.....	19
	Machine-specific user parameters.....	19
1.3	Modified functions.....	20
	NC fundamentals.....	20
	Machining cycles.....	20
	Touch probe cycles (option 17).....	20
	Program functions.....	21
	Programming Q parameters.....	21
	Programming aids.....	21
	Program run.....	21
	Positioning w/ Manual Data Input operating mode.....	21
	Status displays.....	21
	Moving the machine axes.....	21
	Manual touch probe cycles (option 17).....	22
	Tilting the working plane (option 8).....	22
	Tools.....	22

2	Software 81760x-01	23
2.1	New software options	24
	Active Vibration Damping (AVD) (option 146)	24
	Conversational languages	24
2.2	New functions	25
	NC fundamentals	25
	Machining cycles	25
	Touch probe cycles (option 17)	25
	Programming Q parameters	26
	Programming aids	26
	Test run (option 20)	26
	Program run	26
	Active Chatter Control (ACC) (option 145)	27
	Tables	27
	Tools	27
	MOD functions	27
2.3	Modified functions	28
	NC fundamentals	28
	Machining cycles	28
	Touch probe cycles (option 17)	29
	Miscellaneous functions	29
	Programming Q parameters	29
	Programming aids	29
	DXF Converter (option 42)	29
	Moving the machine axes	29
	Manual touch probe cycles (option 17)	29
	Tables	29
	Error messages and help system	30
	MOD functions	30
	Machine-specific user parameters	30

3	Software 81760x-02.....	31
3.1	New software options.....	32
	Extended Tool Management (option 93).....	32
	Renamed software options.....	32
3.2	New functions.....	33
	NC fundamentals.....	33
	Machining cycles.....	33
	Program functions.....	33
	Calling an external NC program.....	33
	Programming Q parameters.....	34
	File management.....	34
	CAD Viewer.....	34
	Manual touch probe cycles (option 17).....	34
	Tables.....	34
	Tools.....	35
	MOD functions.....	35
	HEROS functions.....	35
	Machine-specific user parameters.....	36
3.3	Modified functions.....	37
	NC fundamentals.....	37
	Machining cycles.....	38
	Touch probe cycles (option 17).....	38
	Miscellaneous functions.....	39
	Programming Q parameters.....	39
	Programming aids.....	39
	File management.....	39
	DXF Converter (option 42).....	39
	Program run.....	39
	Status displays.....	39
	Tilting the working plane (option 8).....	40
	Manual touch probe cycles (option 17).....	40
	Tables.....	40
	Tools.....	40
	Machine-specific user parameters.....	40

4	Software 81760x-03.....	41
4.1	New functions.....	42
	Machining cycles.....	42
	Program functions.....	42
	Programming Q parameters.....	42
	Programming aids.....	42
	File management.....	42
	DXF Converter (option 42).....	42
	Program run.....	43
	Pallets (option 22).....	43
	Manual touch probe cycles (option 17).....	43
	Tables.....	43
	Tools.....	43
	Machine-specific user parameters.....	43
4.2	Modified functions.....	44
	NC fundamentals.....	44
	Machining cycles.....	45
	Touch probe cycles (option 17).....	45
	Miscellaneous functions.....	45
	DXF Converter (option 42).....	46
	Program run.....	46
	Pallets (option 22).....	46
	Positioning w/ Manual Data Input operating mode.....	46
	Status displays.....	46
	Manual touch probe cycles (option 17).....	46
	Tables.....	47
	Tools.....	47
	MOD functions.....	47

5	Software 81760x-04.....	49
5.1	New functions.....	50
	NC fundamentals.....	50
	Program functions.....	50
	Programming Q parameters.....	50
	Programming aids.....	51
	Test run (option 20).....	51
	File management.....	51
	Program run.....	51
	Positioning w/ Manual Data Input operating mode.....	52
	Status displays.....	52
	Moving the machine axes.....	52
	Tilting the working plane (option 8).....	52
	Tools.....	52
	Error messages and help system.....	53
	HEROS functions.....	53
	Machine-specific user parameters.....	53
5.2	Modified functions.....	54
	NC fundamentals.....	54
	Machining cycles.....	55
	Touch probe cycles (option 17).....	56
	Program functions.....	56
	Programming Q parameters.....	56
	Calling an external NC program.....	56
	Programming aids.....	56
	Test run (option 20).....	56
	File management.....	57
	DXF Converter (option 42).....	57
	Program run.....	57
	Pallets (option 22).....	57
	Status displays.....	57
	Moving the machine axes.....	57
	Manual touch probe cycles (option 17).....	58
	Tilting the working plane (option 8).....	58
	Tables.....	58
	Tools.....	58
	MOD functions.....	58
	HEROS functions.....	58

6	Software 81760x-05.....	59
6.1	New software options.....	60
	Batch Process Manager (option 154).....	60
	Renamed software options.....	60
6.2	New functions.....	61
	NC fundamentals.....	61
	Machining cycles.....	61
	Touch probe cycles (option 17).....	61
	Program functions.....	61
	Programming Q parameters.....	62
	Calling an external NC program.....	63
	Programming aids.....	63
	Test run (option 20).....	63
	File management.....	63
	CAD Viewer.....	63
	Pallets (option 22).....	63
	Status displays.....	64
	Manual touch probe cycles (option 17).....	64
	Tables.....	64
	Tools.....	64
	Accessories.....	64
	MOD functions.....	65
	HEROS functions.....	65
	Machine-specific user parameters.....	65
6.3	Modified functions.....	66
	NC fundamentals.....	66
	Machining cycles.....	67
	Touch probe cycles (option 17).....	68
	Program functions.....	68
	Miscellaneous functions.....	69
	Programming Q parameters.....	69
	Programming aids.....	69
	Test run (option 20).....	69
	File management.....	69
	Program run.....	70
	Status displays.....	70
	Moving the machine axes.....	70
	Tables.....	70
	Tools.....	70
	Error messages and help system.....	71
	Remote Desktop Manager (option 133).....	71

7	Software 81760x-06	73
7.1	New software options	74
	State Reporting Interface (option 137)	74
	Monitoring of configured machine components for overload (option 155)	74
7.2	New functions	75
	NC fundamentals	75
	Touch probe cycles (option 17)	75
	Program functions	75
	Programming Q parameters	76
	Programming aids	76
	Test run (option 20)	76
	File management	76
	CAD Viewer	77
	Batch Process Manager (option 154)	77
	Program run	77
	Positioning w/ Manual Data Input operating mode	77
	Status displays	77
	Manual touch probe cycles (option 17)	77
	Tilting the working plane (option 8)	78
	Tables	78
	Tools	78
	MOD functions	78
	HEROS functions	78
	Machine-specific user parameters	79
7.3	Modified functions	80
	NC fundamentals	80
	Machining cycles	80
	Touch probe cycles (option 17)	80
	Program functions	81
	Programming Q parameters	81
	Test run (option 20)	81
	CAD Viewer	81
	Moving the machine axes	81
	Manual touch probe cycles (option 17)	81
	Tables	81
	Tools	81
	Remote Desktop Manager (option 133)	82
	HEROS functions	82

8	Software 81760x-07	83
8.1	New software options	84
	HEIDENHAIN OPC UA NC Server (options 56 to 61)	84
	Optimized contour cycles (option 167)	84
	Display step	84
8.2	New functions	85
	Machining cycles	85
	Touch probe cycles (option 17)	85
	Program functions	86
	Miscellaneous functions	86
	Programming Q parameters	86
	Calling an external NC program	86
	Test run (option 20)	87
	Program run	87
	Status displays	87
	Moving the machine axes	87
	Tilting the working plane (option 8)	87
	Tables	87
	Tools	88
	Error messages and help system	88
	HEROS functions	88
	Machine-specific user parameters	88
8.3	Modified functions	89
	Machining cycles	89
	Touch probe cycles (option 17)	89
	Program functions	89
	Miscellaneous functions	89
	Programming Q parameters	90
	Test run (option 20)	90
	File management	90
	Program run	90
	Status displays	90
	Moving the machine axes	90
	Manual touch probe cycles (option 17)	91
	Tables	91
	Tools	91
	MOD functions	91
	HEROS functions	91

9	Software 81760x-08.....	93
9.1	New software options.....	94
	HEIDENHAIN OPC UA NC Server (options 56 to 61).....	94
	Renamed software options.....	94
9.2	New functions.....	95
	NC fundamentals.....	95
	Machining cycles.....	95
	Program functions.....	96
	Programming Q parameters.....	97
	Programming aids.....	97
	Test run (option 20).....	97
	File management.....	97
	CAD Viewer.....	97
	Status displays.....	98
	Tools.....	98
	Error messages and help system.....	98
	MOD functions.....	98
	Remote Desktop Manager (option 133).....	98
	HEROS functions.....	99
	Machine-specific user parameters.....	99
9.3	Modified functions.....	100
	NC fundamentals.....	100
	Machining cycles.....	101
	Touch probe cycles (option 17).....	102
	Miscellaneous functions.....	102
	Programming Q parameters.....	102
	Programming aids.....	102
	Test run (option 20).....	102
	CAD Viewer.....	102
	Batch Process Manager (option 154).....	103
	Program run.....	103
	Pallets (option 22).....	103
	Status displays.....	103
	Moving the machine axes.....	103
	Manual touch probe cycles (option 17).....	103
	Tables.....	104
	Error messages and help system.....	104
	MOD functions.....	104
	Remote Desktop Manager (option 133).....	104
	HEROS functions.....	104
	Machine-specific user parameters.....	104

1

Software
73498x-02

1.1 New software options

Further information: User's Manual for HEIDENHAIN Conversational Programming (ID: 819499-11)

DXF Converter (option 42)

The DXF Converter allows you to open DXF files on the third desktop. You can set a preset and extract contours or machining positions. The control writes the selected elements to an NC program or a point table as **L** and **CC/C** blocks.

Remote Desktop Manager (option 133)

Remote Desktop Manager allows you to use the control to operate an external computer unit (e.g., a Windows PC).

Cross Talk Compensation (CTC) (option 141)

The dynamic acceleration of axes can lead to position errors at the tool tip. **CTC** compensates for these deviations at the tool center point (**TCP**).

Position Adaptive Control (PAC) (option 142)

Depending on the current position, the axes provide different dynamic performance. With **PAC**, the machine parameter adapts the controller parameters depending on the position and speed of an axis.

Load Adaptive Control (LAC) (option 143)

LAC allows the control to automatically determine the workpiece mass, as well as the resulting friction forces. The control dynamically adapts the adaptive feedforward parameters depending on the current mass.

Motion Adaptive Control (MAC) (option 144)

MAC allows the machine manufacturer to adapt the machine parameters depending on motion-dependent input quantities (e.g., speed or following error). This enables improvement of the positioning accuracy at high speeds, for example.

Active Chatter Control (ACC) (option 145)

Chatter marks can be caused during heavy-duty machining, in particular. **ACC** reduces chattering, thereby reducing wear on the tool and machine. In addition, **ACC** increases metal removal rates. In the **ACC** column of the tool table, you activate **ACC** individually for each tool.

1.2 New functions

Further information: User's Manual for HEIDENHAIN Conversational Programming (ID: 819499-11)

Further information: User's Manual for Cycle Programming (ID: 679295-12)

Machining cycles

Cycle **225 ENGRAVING**

This cycle allows you to engrave a text in one plane. You can arrange the texts in a straight line or along an arc.

Touch probe cycles (option 17)

- Cycle **460 CALIBRATION OF TS ON A SPHERE** (ISO: **G460**)
This cycle enables the automatic calibration of a 3-D touch probe using a calibration sphere. You can calibrate only the radius or you can calibrate both the radius and the length of the touch probe.
- Cycle **461 TS CALIBRATION OF TOOL LENGTH** (ISO: **G461**)
This cycles allows you to calibrate the length of a touch probe using a calibration ring.
- Cycle **462 CALIBRATION OF A TS IN A RING** (ISO: **G462**)
This cycle allows you to calibrate the radius of a touch probe using a calibration ring. The cycle optionally also determines the center offset of the touch probe.
- Cycle **463 TS CALIBRATION ON STUD** (ISO: **G463**)
This cycle allows you to calibrate the radius of a touch probe using a calibration mandrel. The cycle optionally also determines the center offset of the touch probe.
- Cycle **484 CALIBRATE IR TT** (ISO: **G484**)
This semiautomatic cycle allows you to calibrate the wireless TT 449 tool touch probe using a calibration mandrel. You need to recalibrate the tool touch probe if you change its position.
- Cycle **452 PRESET COMPENSATION** (ISO: **G452**, option 48)
This cycle enables the compensation of drift or the adjustment of multiple interchangeable heads of the machine. This requires a previous reference measurement using Cycle **451 MEASURE KINEMATICS** (ISO: **G451**)

Miscellaneous functions

M197 allows you to avoid the rounding of outside contour corners at high contour speeds.

Program functions

Many cycles use the same cycle parameters (e.g., **Q208 RETRACTION FEED RATE**). In the NC program, you can define default values for some of these parameters.

The following functions are available:

- The **GLOBAL DEF** function allows you to define default values for various cycle groups (e.g., probing).
- Press the **SET STANDARD VALUES** soft key to load the default values into the cycle. The control then enters the word **PREDEF** in the cycle definition.

Programming Q parameters

Freely definable tables (*.tab) allow you to create your own tables for defining measuring positions, for example. You can access the table from within the NC program.

The following functions are available:

- The **EDIT FORMAT** soft key allows you to modify the table format.
- Use the function **FN 26: TABOPEN (ISO: D26)** to open a freely definable table so that you can work with it. Only one table can be opened in an NC program at any one time.
- The function **FN 27: TABWRITE (ISO: D27)** allows you to write to the numerical fields of the open table. You can write to no more than eight columns within one NC block. In a Q parameter, you define the value to be written.
- The function **FN 28: TABREAD (ISO: D28)** allows you to read the numerical fields of the open table and save the values to successive Q parameters. In one line, you can read the values from a maximum of eight columns.

File management

You can configure a DHCP/DNS server for all Ethernet interfaces except the primary interface. This makes it possible to supply devices in the network with dynamic IP addresses and device names. You configure the server in the network dialog of the file manager.

Moving the machine axes

On machines with a rotary head axis, you can activate the axis position of the tool axis as a virtual tool axis **VT**. You can activate the virtual tool axis in **Manual operation** mode and during handwheel superimpositioning (option 21). The machine manufacturer can also permanently assign the virtual tool axis to an axis key on the handwheel.

Manual touch probe cycles (option 17)

The manual touch probe cycle **Probe centerline** allows you to set a preset in the center of a ridge or slot.

Accessories

- The control supports the HR 520 and HR 550 FS handwheels.
The handwheels feature a display that shows information from the control (e.g., axis positions). You can use the handwheel soft keys for setup functions (e.g., the setting of presets).
- The control supports the wireless TT 449 tool touch probe that allows you to measure tools in the machine.

MOD functions

- The **External access** MOD function in the **Machine settings** group allows you to block network accesses to the control.
- The **SET UP WIRELESS HANDWHEEL** soft key in the MOD menu allows you to configure the HR 550 FS wireless handwheel.

Machine-specific user parameters

- The optional machine parameter **kinemAtStartup** (no. 204002) allows you to define the kinematic model to be activated by the control during booting. This setting is independent of the current kinematics.
- The machine parameter **operatingTimeReset** (no. 200801) allows you to define whether the control resets the machining time at program start.

1.3 Modified functions

Further information: User's Manual for HEIDENHAIN Conversational Programming (ID: 819499-11)

Further information: User's Manual for Cycle Programming (ID: 679295-12)

NC fundamentals

- The maximum number of characters for the tool name in **TOOL DEF** blocks (ISO: **T**) or **TOOL CALL** blocks (ISO: **G51**) has been increased from 16 to 32.
- When you copy a subprogram in the NC program, the control checks the LBL name. If the LBL name has already been assigned, the control displays a warning.
- When you copy a block in the NC program, the control automatically adapts any block references contained. As a result, for example, the references of an **FK** contour (option 19) are retained.

Machining cycles

- The parameter **Q437 APPROACH POSITION** has been added to Cycle **256 RECTANGULAR STUD** (ISO: **G256**, option 19). This parameter allows you to define whether the control approaches the stud along the straight line on the right or at one of the corners.
- The parameter **Q376 STARTING ANGLE** has been added to Cycle **257 CIRCULAR STUD** (ISO: **G257**, option 19). You define the angle at which the control starts machining. If you define a value of -1, then the control automatically calculates the position.
- Cycle **25 CONTOUR TRAIN** (ISO: **G125**, option 19) supports the **APPR** and **DEP** functions in the contour description.
- You can call the SL Cycles (option 19) even if radius compensation is active.

Touch probe cycles (option 17)

- After optimizing the angular orientation with Cycle **451 MEASURE KINEMATICS** (ISO: **G451**, option 48), a separate measurement series for position optimization is no longer required.
- The input range of parameter **Q423 NO. OF PROBE POINTS** has been increased from a range of 3 to 4 points to a range of 3 to 8 points in Cycle **451 MEASURE KINEMATICS** (ISO: **G451**, option 48).
- Cycle **451 MEASURE KINEMATICS** (ISO: **G451**, option 48) allows you to measure the offset of the first rotary head axis. For this purpose, you run the cycle using two styli of different lengths. The control saves the result to the Q parameters **Q147** to **Q149**.

Program functions

When programming the **PLANE** functions (option 8), you can also define **SEQ** by means of a Q parameter. If a value of 0 is entered in the Q parameter, then **SEQ** will not be defined. The values of +1 and -1 mean **SEQ+** and **SEQ-**, respectively.

Programming Q parameters

The text file defining the output format of **FN 16: F-PRINT** (ISO: **D16**) may also contain lower-cased code words. The control does not differentiate between **M_CLOSE**, **M_Close**, and **m_close**, for example.

Programming aids

- You can activate the calculator from within any operating mode. It will also remain active when the operating mode is switched.
- You can move the calculator using the cursor keys or the mouse.
- The calculator provides three modes: **Standard**, **Scientific**, and **Equation editor**. Depending on the view, the functions available vary.
- The **AX. VALUES** soft key allows you to load the current axis position and use it in calculations.

Program run

When running a pallet table (option 22) in **Program run, single block** or **Program run, full sequence** operating mode, the table is highlighted in white instead of in yellow.

Positioning w/ Manual Data Input operating mode

If an NC block contains a tool radius compensation in **Positioning w/ Manual Data Input** operating mode, then the control issues an error message.

Status displays

- If you use a spindle as a C axis, then the control displays the axis letter **C** instead of **S2** in the distance-to-go display of this spindle.
- In **Test Run** operating mode, the control does not display the **TT** tab of the additional status display, because it does not have any effect during simulation.
- The control shows the active kinematic model on the **POS** tab of the additional status display in all operating modes.
- The tool type graphics on the **TOOL** tab of the additional status display have been updated.

Moving the machine axes

You can use handwheel superimpositioning with **M118** (option 21) when **FUNCTION TCPM** (option 9) is active.

Manual touch probe cycles (option 17)

- You can also use manual touch probe cycles in conjunction with mechanical probes or milling cutters. The **ACTUAL POSITION CAPTURE** soft key replaces the touch probe signal upon deflection of the stylus. This allows you to use the manual touch probe cycles without the **Touch Probe Functions** software option (option 17).
- In the manual touch probe cycles, instead of individual positions, you can also define a hole or a circular stud that is automatically probed by the control.

The function is available in the following touch probe cycles:

- **Probing of rotation**
- **Corner as preset**
- **Circle probing**
- **Probe centerline**
- The manual touch probe cycles check for whether the current angle of the rotary axes matches the active tilt angle. If this is not the case, the control issues an error message.
- The **ALIGN ROT. TABLE** soft key has been added to the manual touch probe cycle **Probing of rotation**. This soft key allows you to compensate for workpiece misalignment by rotating the table.

Tilting the working plane (option 8)

You can define up to three tilt angles in the 3-D ROT menu. You define the rotation using the spatial angles **SPA**, **SPB**, and **SPC**.

Tools

The maximum number of characters permitted in the **NAME** and **DOC** columns of the tool table has been increased from 16 to 32.

2

Software 81760x-01

2.1 New software options

Further information: User's Manual for HEIDENHAIN Conversational Programming (ID: 1096883-10)

Active Vibration Damping (AVD) (option 146)

AVD dampens machine vibrations arising during machining, thereby improving the workpiece surface quality.

Conversational languages

The **Additional Language** software option (option 41) is no longer available.

The following conversational languages are also available without the option 41:

- Slovenian
- Norwegian
- Slovak
- Korean
- Turkish
- Romanian

The conversational languages Latvian, Estonian, and Lithuanian are no longer available.

2.2 New functions

Further information: User's Manual for HEIDENHAIN Conversational Programming (ID: 1096883-10)

Further information: User's Manual for Cycle Programming (ID: 1096886-10)

NC fundamentals

- The **BLK FORM CYLINDER** function allows you to define a cylindrical workpiece blank by entering the radius, length, and an optional oversize. If you additionally enter an inside radius, then you define a tube.
- The **BLK FORM ROTATION** function allows you to define a rotationally symmetric workpiece blank. You program the workpiece blank contour in a subprogram and call the subprogram in the workpiece blank definition.
- In a **TOOL CALL** block (ISO: **T**), you can enter a QS parameter instead of a tool name.
- In a **TOOL CALL** block (ISO: **T**), you can select the tool from a pop-up window. The control marks all tools available in the tool magazine green.
- The **APPR** and **DEP** functions for approach and departure are available in ISO programming mode.

Machining cycles

- Cycle **233 FACE MILLING** (ISO: **G233**, option 19)
This cycle enables the face milling of a surface along meandering, linear, or helical paths. In order to machine shoulders, you can optionally define up to three limits.
- Cycle **275 TROCHOIDAL SLOT** (ISO: **G275**, option 19)
This cycle allows you to machine open and closed slots using a trochoidal milling process. You define the center path for the slot in a subprogram and the width of the slot in the cycle parameters.

Touch probe cycles (option 17)

Cycle 4 MEASURING IN 3-D

This cycle allows you to determine any position on the workpiece. You define the direction of approach in the cycle by using a vector. The control saves the measured coordinates to successive Q parameters. If the control is not able to determine a point, then the 4th Q parameter is assigned a value of -1.

Programming Q parameters

- The **FN 9** function allows you to define an NC program jump that is linked to a condition.

The following functions are available in addition to the comparison with a value:

- **IS DEFINED** causes the jump to be made at a defined Q parameter.
- **IS UNDEFINED** causes the jump to be made at an undefined Q parameter.
- The functions of **FN 18** (ISO: **D18**) have been enhanced:
 - In **FN 18: SYSREAD (D18) ID50** and **ID51**, you can use **IDX** to define indexed tools.
 - **FN 18: SYSREAD (D18) ID50**: values in the tool table
 - **NR32**: value in the **T-ANGLE** column
 - **NR34**: value in the **LIFTOFF** column
 - **NR35**: value in the **R2TOL** column
 - **NR37**: touch probe table line for the tool
 - **NR38**: time stamp of last usage

Programming aids

- The **PROGRAMMING AIDS** soft key has been added to the soft-key row for special functions. This soft key opens a menu where you can add a comment or a structure item to the NC program.
- The control provides a cutting data calculator for calculating the spindle speed and the feed rate.

The **CUTTING DATA CALCULATOR** soft key is available while the calculator is open and while entering the values for the feed rate or spindle speed. You can load the results directly into the dialog.

Test run (option 20)

- The control provides a high-resolution 3-D motion simulation feature.
- You can choose between a view of the solid object, a view of the tool paths, or a view combining these two views.
- View options, such as for displaying a transparent workpiece and tool, highlighting the workpiece edges, or displaying the workpiece in color, allow you to adapt the simulation to your specific requirements.
- The soft keys for setting the simulation speed have been redesigned.
- The selected simulation speed is retained until power is interrupted.
- The simulation considers the columns **R2**, **LCUTS**, and **T-ANGLE** of the active tool table.

Program run

The **Retraction** operating mode allows you to disengage the tool from the workpiece after an interruption in power. The machine manufacturer defines the scope of the **Retraction** operating mode.

Active Chatter Control (ACC) (option 145)

- You can activate and deactivate **ACC** by soft key in the operating modes **Positioning w/ Manual Data Input**, **Program run, single block**, and **Program run, full sequence**.
- When **ACC** is active, the control displays a corresponding icon in the general status display.

Tables

If a table column is assigned the **UPTEXT** column type, then the control will convert the entered text to uppercase.

Tools

The **PITCH** column, in which you define the pitch of a thread-cutting tool (e.g., a tap), has been added to the tool table.

MOD functions

- The **Graphic settings** group that allows you to modify the 3-D motion simulation has been added to the MOD menu. You can choose between the available model types (3-D model, 2.5-D model, and no model) and set the model quality.
- The **Show the info line** function has been added to the **Display settings** group. While this function is inactive, the information line will not be shown in the operating modes. The control will then divide long error messages into two lines.
- The machine-dependent function **Tool-usage file** has been added to the **Machine settings** group. You select when the control should generate a tool usage file.
- The **Set the system time** function has been added to the **System settings** group. You can manually set the time zone, date, and time, or have it synchronized via an NTP server.

2.3 Modified functions

Further information: User's Manual for HEIDENHAIN Conversational Programming (ID: 1096883-10)

Further information: User's Manual for Cycle Programming (ID: 1096886-10)

NC fundamentals

- If you use the **ACTUAL POSITON CAPTURE** key to create a linear block containing the current axis positions, then the linear block contains only the axes of the active kinematic model.
- While running a program in **Programming** operating mode, you can simultaneously copy and paste blocks in the NC program.

Machining cycles

- The German umlauts and the diameter symbol have been added to the character set of Cycle **225 ENGRAVING** (ISO: **G225**).
- In Cycle **22 ROUGH-OUT** (ISO: **G122**, option 19), the input range of the parameter **Q18 COARSE ROUGHING TOOL** has been increased to +99 999.
- The parameter **Q395 DEPTH REFERENCE** has been added to Cycles **200 DRILLING** (ISO: **G200**), **203 UNIVERSAL DRILLING** (ISO: **G203**), and **205 UNIVERSAL PECKING** (ISO: **G205**, option 19). This parameter allows you to define whether the cycle takes into account the **T-ANGLE** column of the tool table in the depth.
- The parameter **Q208 RETRACTION FEED RATE** has been added to Cycle **205 UNIVERSAL PECKING** (ISO: **G205**, option 19). This parameter allows you to define the feed rate at which the tool retracts from the hole at the end of the machining operation.
- Cycle **241 SINGLE-LIP D.H.DRLNG** (ISO: **G241**, option 19) has been modified to reduce the risk of tool breakage.

The following parameters have been added:

- **Q435 DWELL DEPTH**
- **Q401 FEED RATE FACTOR**
- **Q202 MAX. PLUNGING DEPTH**
- **Q212 DECREMENT**
- **Q205 MIN. PLUNGING DEPTH**
- The input range of parameter **Q256 DIST FOR CHIP BRKNG** has been increased to a maximum of three decimal places in some cycles for drilling and thread machining (option 19).
- The parameter **Q512 FEED FOR APPROACH** has been added to Cycles **262 THREAD MILLING** (ISO: **G262**), **263 THREAD MLLNG/CNTSNKG** (ISO: **G263**), **264 THREAD DRILLNG/MLLNG** (ISO: **G264**), and **267 OUTSIDE THREAD MLLNG** (ISO: **G267**, option 19). When using a small tool, you can reduce the risk of tool breakage by approaching at a low feed rate.
- Cycles **206 TAPPING** (ISO: **G206**), **207 RIGID TAPPING** (ISO: **G207**), and **209 TAPPING W/ CHIP BRKG** (ISO: **G209**, option 19) check for whether the actual pitch matches the pitch defined in the **PITCH** column of the tool table.

Touch probe cycles (option 17)

The parameter **Q305 NUMBER IN TABLE** has been added to Cycle **404 SET BASIC ROTATION** (ISO: **G404**). You can immediately activate the basic rotation or save it to a line of the preset table.

Miscellaneous functions

You can program up to four miscellaneous functions in a separate M block.

Programming Q parameters

When you move the output window of function **FN 16: F-PRINT** (ISO: **D16**), the control saves the new position.

Programming aids

The control also displays the block numbers of circles and circular arcs in the programming graphics.

DXF Converter (option 42)

- You can add folders to bookmarks within the file manager of the DXF Converter.
- If option 42 is active, then, by default, the control displays DXF files in the file manager.

Moving the machine axes

After an NC stop with **LIFTOFF** active, you need to press the **MANUAL TRAVERSE** soft key so that you will again be able to use the handwheel for positioning.

Manual touch probe cycles (option 17)

- You can use the manual touch probe cycle **Probe centerline** to determine the centerline between two holes or studs. At the end of the cycle, you select the axis for which the control should load the result.
- If the first measuring point you have defined in the manual touch probe cycle **Probe centerline** is the center of a hole or a stud, then you can use a workpiece edge in any direction to define the second measuring point.
- If you manually calibrate a touch probe, you need to confirm the calibration data by pressing the **OK** soft key. After you have confirmed the new data, the control hides the soft key and outputs the log.

Tables

- When you import a tool table from the iTNC 530, the control loads all supported tool types and assigns the tool type **Milling cutter,MILL** to all unknown tool types.
- You can import datum tables (*.d), pocket tables (*.tch), preset tables (*.pr), and point tables (*.pnt) from the iTNC 530.

Error messages and help system

The format and layout of the keystroke log that you can call in the error window has been modified to make it easier to read.

MOD functions

In the **Display settings** MOD group, you select which information the control is to show in the position display of the general and additional status displays.

The following options have been added:

- **ACTDS**: the position display shows the distance to go in the input coordinate system.
- **REFDST**: the position display shows the distance to go in the machine coordinate system
- **M118**: the position display shows the traverse paths performed with handwheel superimpositioning (option 21).

Machine-specific user parameters

- The control supports searching for language-specific names.
- Certain machine parameters are available only after you have enabled a software option. If an option has been enabled only temporarily, the control disables the parameters after expiry of the option period. Parameters that still contain values are marked by the control with a red x. You can clear the parameter values, but you cannot edit them.
- The input range of the parameter **maxLineCommandSrch** (no. 105412) has been changed from 9999 lines to 50 000 lines. The default value has been changed from 500 lines to 20 00010 000 lines.

3

Software 81760x-02

3.1 New software options

Further information: User's Manual for HEIDENHAIN Conversational Programming (ID: 1096883-11)

Extended Tool Management (option 93)

The extended tool management provides the **Tooling list** and **T usage order** tabs. After a tool usage test, the tools of the active NC program and their sequence are shown on these tabs.

Renamed software options

The following software options have been renamed:

- The option 8 is now called **Advanced Function Set 1**
- The option 9 is now called **Advanced Function Set 2**
- The option 21 is now called **Advanced Function Set 3**

3.2 New functions

Further information: User's Manual for HEIDENHAIN Conversational Programming (ID: 1096883-11)

Further information: User's Manual for Cycle Programming (ID: 1096886-11)

NC fundamentals

When you program in ISO programming mode using the alphabetic keyboard, the control automatically uppercases the first letter of an NC block.

Machining cycles

- Cycle **270 CONTOUR TRAIN DATA** (ISO: **G270**, option 19)
This cycle allows you to add a type of approach, type of departure, and radius compensation to the data provided by Cycle **25 CONTOUR TRAIN** (ISO: **G125**). You need to define Cycle **270 (G270)** before Cycle **25 (G125)**.
- Cycle **39 CYL. SURFACE CONTOUR** (ISO: **G139**, option 8, option 19)
This cycle allows you to mill a contour on a cylindrical surface. You define the contour in a subprogram on the unrolled cylindrical surface.
- Cycle **239 ASCERTAIN THE LOAD** (ISO: **G239**, option 143)
This cycle allows the control to perform a weighing run and ascertain the current table load. Load Adaptive Control (**LAC**) enables the control to adapt specific feedforward and controller parameters to the current load. The machine manufacturer defines the movements to be performed by the control during the weighing run.

Program functions

- **FUNCTION FEED DWELL** allows you to program a cyclic dwell time. Use the parameters **D-TIME** and **F-TIME** to define the dwell time and the machining time in seconds.
- You can also orient the working plane perpendicularly to the tool without defining rotary axes (e.g., when using a mounted angle head). Use the **PLANE SPATIAL** function (option 8) and the **STAY** positioning behavior to tilt the working plane to the exact angle of the angle head.

Calling an external NC program

The **PGM CALL** function (ISO: %) has been enhanced as follows:

- The **SELECT PROGRAM** soft key allows you to define an NC program that can be called later.
- The **CALL SELECTED PROGRAM** soft key allows you to call the previously defined NC program at this location.

Programming Q parameters

The **FN 18: SYSREAD** functions (ISO: **D18**) have been enhanced:

- **FN 18: SYSREAD (D18) ID52**
 - **NR1:** pocket number of a tool
 - **NR2:** magazine number of a tool
- **FN 18: SYSREAD (D18) ID71 NR0:** axis for which the **LAC** weighing run was last performed
- **FN 18: SYSREAD (D18) ID320 NR1:** current system time in seconds since 1970-01-01 00:00:00 UTC
- **FN 18: SYSREAD (D18) ID990 NR8:** current spindle angle
- **FN 18: SYSREAD (D18) ID992**
 - **NR31:** check for whether radius compensation is possible in **Positioning w/ Manual Data Input** operating mode
 - **NR32:** check for whether a cycle can be called
- **FN 18: SYSREAD (D18) ID1020 NR13:** check for whether a configuration subfile has been loaded

File management

You can select and edit HEIDENHAIN unit programs (*.hu) and HEIDENHAIN contour programs (*.hc) in any operating mode.

CAD Viewer

The **CAD-Viewer** allows you to open CAD files on the control without the need for option 42. The **CAD-Viewer** supports STEP, IGES, and DXF files.

In the **CAD-Viewer**, you can choose among different views and set presets. When hovering the mouse pointer over the model, the **CAD-Viewer** displays the coordinates of the mouse pointer relative to the preset.

Manual touch probe cycles (option 17)

The manual touch probe cycle **Probe in plane** allows you to measure the misalignment of a plane. After you have probed three points on the plane, the control will calculate the 3-D basic rotation from these points.

You can save the 3-D basic rotation to the preset table, activate it, or use the **ALIGN ROT. AXES** soft key to align the rotary axes relative to the 3-D basic rotation.

If a 3-D basic rotation has been saved to the active preset, the control displays a corresponding icon in the general status display.

Tables

The **LOCKED** column of the preset table allows you to activate write protection for each line.

You can also lock the line with a password. When a line is locked, the control displays its values in red.

Tools

- The **APPEND N LINES AT END** soft key has been added to tool management. The soft key allows you to append as many lines as you need to the end of the table.
- The control provides additional tool types to increase compatibility with the iTNC 530:
 - **Undefined,******
 - **Boring tool,BOR**
 - **Back boring tool,BCKBOR**
 - **Thread mill,GF**
 - **Thread mill w/ countersink,GSF**
 - **Thread mill w/ single thread,EP**
 - **Thread mill w/ indxbl insert,WSP**
 - **Thread milling drill,BGF**
 - **Circular thread mill,ZBGF**
 - **Roughing cutter,MILL_R**
 - **Finishing cutter,MILL_F**
 - **Rough/finish cutter,MILL_RF**
 - **Floor finisher,MILL_FD**
 - **Side finisher,MILL_FS**
 - **Face milling cutter,MILL_FACE**

MOD functions

- The **Traverse limits** MOD function has been added to the **Machine settings** group. You can set an upper and a lower limit for each axis in order to protect machine components, for example.
- The machine manufacturer can disable the kinematics selection for simulation.

HEROS functions

- The option **Allow USB storage devices** has been added to the **SELinux** settings in the HEROS menu. If you deactivate this option, the control will not be able to connect USB data media. Instead, the control displays a warning for fifteen seconds.
- The control provides a media player. If the machine manufacturer has configured the media player, you can open audio and video files in the OGG format (*.ogg, *.oga, *.ogv, *.ogx).

Machine-specific user parameters

- The machine parameter **posAfterContPocket** (no. 201007) allows you to define how the control moves to clearance height at the end of an SL cycle.
- The machine parameter **CfgTT** (no. 122700) for configuring the tool touch probe has been enhanced:
 - **stopOnCheck** (no. 122717): response of the control if the breakage tolerance is exceeded during tool inspection
 - **stopOnMeasurement** (no. 122718): response of the control if the breakage tolerance is exceeded during tool measurement
 - **adaptToolTable** (no. 122719): change made to the tool table after inspection, after measurement, or never
- The optional machine parameter **progToolCallDL** (no. 124501) allows you to define how the control interprets the parameter **DL** from the tool call. The machine manufacturer enables this parameter.

3.3 Modified functions

Further information: User's Manual for HEIDENHAIN Conversational Programming (ID: 1096883-11)

Further information: User's Manual for Cycle Programming (ID: 1096886-11)

NC fundamentals

- The dialog for shutting down the control has been modified. You define whether the control shuts down or restarts.
- In the tool call, you can define a feed rate per tooth (**FZ**) or a feed rate per revolution (**FU**).
- Press the **GOTO** key to select a table line in the pop-up window for tool selection.
- When you import an NC program from a previous control model, the control no longer generates ERROR blocks for functions that are not supported. Instead, when testing or running the NC program, the control interrupts the test run or program run with an error message.

This change applies to the following functions:

- Miscellaneous functions: **M104**, **M105**, **M112**, **M114**, **M124**, **M134**, **M142**, **M150**, and **M200** to **M204**
- Commands: **FN 15** (ISO: **D15**), **FN 31** (ISO: **D31**), **FN 32** (ISO: **D32**), **FT**, and **FMAXT**

Machining cycles

- The character set provided by Cycle **225 ENGRAVING** (ISO: **G225**) has been expanded. You can engrave the CE mark, the letter **ß**, and the symbol **@**. In addition, you can program the current date and the system time using a system variable.
- Cycles **252 CIRCULAR POCKET** (ISO: **G252**), **253 SLOT MILLING** (ISO: **G253**), and **254 CIRCULAR SLOT** (ISO: **G254**, option 19) have been enhanced:
 - The parameter **Q439 FEED RATE REFERENCE** allows you to define whether the programmed feed rate is referenced to the cutting edge of the tool or the path of the tool center.
 - To prevent residual material, the cycles take into account the column **R2** of the tool table during finishing.
- Cycles **256 RECTANGULAR STUD** (ISO: **G256**) and **257 CIRCULAR STUD** (ISO: **G257**, option 19) maintain a constant lateral safety clearance of 2 mm during approach and departure.
- If the parameter **Q389 MILLING STRATEGY** is defined with a value of 4 in Cycle **233 FACE MILLING** (ISO: **G233**, option 19), then the control also considers the parameter **Q370 TOOL PATH OVERLAP** during the first infeed.
- Cycle **22 ROUGH-OUT** (ISO: **G122**, option 19) has been enhanced.
The following parameters have been added:
 - **Q401 FEED RATE FACTOR**: percentage feed rate factor with the tool being in full contact with the workpiece
 - **Q404 FINE ROUGH STRATEGY**: traversing strategy for fine roughing with a smaller tool

Touch probe cycles (option 17)

- The input range of parameter **Q305 NUMBER IN TABLE** has been extended from +2999 to +99 999 in touch probe cycles.
- The parameter **Q536 STOP BEFORE RUNNING** has been added to Cycle **484 CALIBRATE IR TT** (ISO: **G484**). This parameter allows you to define whether the cycle is run semi-automatically or fully automatically.
- The input range of parameter **Q409 MEMORY DESIGNATION** has been increased to +99 999 in Cycle **450 SAVE KINEMATICS** (ISO: **G450**, option 48).
- The control displays a pop-up window in Cycles **451 MEASURE KINEMATICS** (ISO: **G451**) and **452 PRESET COMPENSATION** (ISO: **G452**, option 48). If the distance to go is less than the ball tip radius, then this window will be displayed continuously.
- During the KinematicsOpt cycles (option 48) the control no longer traverses at **FMAX**. Instead, it traverses at the touch probe's rapid traverse.

Miscellaneous functions

The function **M116** (option 8) causes the control to interpret the feed rate of a rotary axis as mm/min or 1/10 inches/min instead of as degrees/min.

M116 is also effective for axes that have been deselected by means of **M138**, **M128**, or **FUNCTION TCPM** (option 9).

Programming Q parameters

- The maximum size of files output with **FN 16: F-PRINT** (ISO: **D16**) has been increased from 4 kB to 20 kB.
- The arrangement of the soft keys for programming the SQL commands has been modified.

Programming aids

The orange polar coordinate key **P** of the calculator allows you to insert the number pi.

File management

- The **COPY FIELD** soft key has been added to the **Last files** pop-up window. This soft key allows you to save the path of the marked file to the clipboard.
- When you copy large files from the control to a USB device, the control displays the transmission progress.
- The control displays the message **The USB device can be removed now** to let you know when you can safely remove the USB device.

DXF Converter (option 42)

The user interfaces of the DXF Converter and the **CAD-Viewer** have been standardized.

Program run

The behavior of the control in **Program run, single block** operating mode has been modified. When executing point pattern cycles and the function **CYCL CALL PAT** (ISO: **G79 PAT**), the control moves to clearance height after every position and stops machining.

Status displays

- The basic rotation icon has been modified to match the 3-D basic rotation icon.
- The **control-in-operation** icon is shown on the screen as long as a positioning command for the axes is pending. The control displays the icon in the general status display even if the feed rate override is 0 %.
- In **Manual Operation** mode, the control shows the contouring feed rate in the general status display.
- The **Q PARAMETER LIST** soft key allows you to define the Q parameters displayed by the control on the **QPARA** tab of the additional status display. The input range of this dialog has been increased to 132 characters per line.

Tilting the working plane (option 8)

The **PLANE RESET** function does not affect Manual operation mode. To deactivate an active tilting function in Manual operation mode, use the 3-D ROT menu.

Manual touch probe cycles (option 17)

- The number of positioning movements has been reduced in the manual calibration cycle **Calibration of a TS in a ring**. The **Safety clearance?** input parameter has been removed.
- In the manual calibration cycle **TS calibration of tool length**, you can select only the active tool axis as the probing direction.

Tables

- The input range of the **DOC** column of the pocket table has been extended to 32 characters.
- The preset table is write-protected in **Programming** operating mode. You can call the table, but you cannot edit it.

Tools

In tool management, you can use the **GOTO** key to move to a table line.

Machine-specific user parameters

The control checks an NC program for errors when it is opened. The machine parameter **maxLineGeoSrch** (no. 105408) defines up to which line the control should check the NC program. The input range of this parameter has been increased from 9999 lines to 50 000 lines. The default value has been changed from 100 lines to 10 000 lines.

4

Software 81760x-03

4.1 New functions

Further information: User's Manual for HEIDENHAIN Conversational Programming (ID: 1096883-12)

Further information: User's Manual for Cycle Programming (ID: 1096886-12)

Machining cycles

Cycle **258 POLYGON STUD** (ISO: **G258**, option 19)

This cycle allows you to machine a stud with any number of corners. You define the stud in the cycle parameters.

Program functions

- **FUNCTION S-PULSE** allows you to program a pulsing spindle speed (e.g., to avoid self-oscillations of the machine). The **SCALE** parameter defines the percentage by which the spindle speed may vary within the period of time defined in **P-TIME**.
- The **SELECT ANIMATION** soft key has been added to the soft-key row for the **PLANE** functions (option 8). When you press this soft key, the control shows an animation of the tilting strategy for the selected tilting function.

Programming Q parameters

The **FN 18: SYSREAD** (ISO: **D18**) functions have been enhanced:

- **FN 18: SYSREAD (D18) ID56**
 - **NR1:** number of lines in the active tool table
 - **NR2:** number of lines in the active datum table
 - **NR4:** number of lines in the open, freely definable table
- **FN 18: SYSREAD (D18) ID360 NR11:** if the error message is suppressed, check whether the probing operation has been completed successfully or the touch point has not been reached

Programming aids

In **Programming** operating mode, the screen layout **PROGRAM + SECTS** allows you to edit the structure items in the structure window.

File management

- You can search for files within the current folder by entering the file name using the alphabetic keyboard. After you have entered the first character, the control opens an input window containing the search criterion and highlights the first file whose name matches it.
- The control shows the name of the file in a separate line beneath the displayed path.

DXF Converter (option 42)

The DXF Converter can also generate a circular arc **CR**. You can choose whether you want the DXF Converter to output **CC** and **C** circular arcs or **CR** circular arcs.

Program run

The control differentiates between an interrupted and a stopped NC program.

When the control is in an interrupted state, it provides the following options:

- Switching the operating mode
- Manual traverse
- Changes to Q parameters
- Changes to the settings for skipping NC blocks
- Changes to the settings for program interruptions with **M1**

When the NC program is in an interrupted state, the **control-in-operation** icon is shown in transparent red in the general status display.

Pallets (option 22)

In the pallet table, the **SELECT** soft key allows you to select the values of the **NAME** and **DATUM** columns.

Manual touch probe cycles (option 17)

You can transfer the values measured in a manual touch probe cycle into the preset table by defining a line number in the **Number in table?** input field.

If the line number defined does not exist, the control creates a new line. If the selected line is protected with a password, you need to enter the password.

Tables

- The **PATHNAME** column type for tables has been added. For this type of column, you can use the **SELECT** soft key to select a file path from a pop-up window.
- Tool tables to be imported might contain columns that are not supported by the control. When importing such columns, the control displays a message informing you that the values will not be transferred.

Tools

- The control takes into account tool carriers. The additional **ToolHolderWizard** tool allows you to create tool carriers or parameterize available 3-D models.
- In the **KINEMATIC** column of the tool table, you use the **SELECT** soft key to assign a tool-carrier kinematic model to a tool.

Machine-specific user parameters

The optional machine parameter **restoreCoolant** (no. 201008) allows you to define whether the control restores the status of the cooling lubrication (**M7** or **M8**) after the execution of Cycles **202 BORING** (ISO: **G202**) and **204 BACK BORING** (ISO: **G204**, option 19).

4.2 Modified functions

Further information: User's Manual for HEIDENHAIN Conversational Programming (ID: 1096883-12)

Further information: User's Manual for Cycle Programming (ID: 1096886-12)

NC fundamentals

- The functions **APPR LT**, **APPR LCT**, **DEP LT**, and **DEP LCT** position all of the required axes simultaneously to an auxiliary point.
- The functions **APPR CT** and **DEP CT** allow you to approach and depart a helix. The programmed helix is extended by adding the approach motion and the departure motion.
- The soft keys of the functions for approach and departure are arranged on the same soft-key row.
- During the editing of a marked NC block, the marking is retained.
- If, when editing a marked NC block, you go to an identical syntax element of another NC block using the cursor keys, then the control also marks all NC blocks between these two blocks.
- When the **PROGRAM + GRAPHICS** screen layout is selected, the soft-key row of the **FK** functions (option 19) provides the **START** and **START SINGLE** soft keys. On a 19-inch screen, the soft-key row also provides the **RESET + START** soft key.

Machining cycles

- A mirror image that was programmed using Cycle **8 MIRRORING** (ISO: **G28**) is also effective for a subsequent tilting of the working plane programmed with a **PLANE** function (option 8). The control mirrors only the axis positions, and not the tilt angle.
- During the execution of Cycle **251 RECTANGULAR POCKET** (ISO: **G251**, option 19), the control verifies that the active tool radius, including the programmed oversize, is smaller than the corner radius defined in the cycle.
- You can call the following cycles with the functions **CYCL CALL PAT** (ISO: **G79 PAT**) and **CYCL CALL POS** (ISO: **G79 | G01**):
 - Cycle **232 FACE MILLING** (ISO: **G232**, option 19)
 - Cycle **233 FACE MILLING** (ISO: **G233**, option 19)
- The SL Cycles (option 19) take into account the function **M110** so that, on inside radii, the feed rate remains constant at the cutting edge.
- If you define the parameter **Q210 DWELL TIME AT TOP** in Cycles **200 DRILLING** (ISO: **G200**) and **203 UNIVERSAL DRILLING** (ISO: **G203**, option 19), then the control positions the tool to the value defined in the parameter **Q200 SET-UP CLEARANCE** during the dwell time.
- In Cycle **205 UNIVERSAL PECKING** (ISO: **G205**, option 19), the control positions the tool to the value defined in parameter **Q200 SET-UP CLEARANCE** for chip removal.
- In Cycles **202 BORING** (ISO: **G202**) and **204 BACK BORING** (ISO: **G204**, option 19), the control positions the tool to the center of the hole at the end of the machining operation. You can also use incremental coordinates to program the machining positions.
- Within Cycle **247 PRESETTING** (ISO: **G247**), you can select the preset from the preset table by soft key.

Touch probe cycles (option 17)

- Cycles **421 MEASURE HOLE** (ISO: **G421**), **422 MEAS. CIRCLE OUTSIDE** (ISO: **G422**), and **427 MEASURE COORDINATE** (ISO: **G427**) have been enhanced for the automatic compensation of turning tools.

The following parameters have been added:

- **Q498 REVERSE TOOL**: tool rotated by 180° during machining
- **Q531 ANGLE OF INCIDENCE**: inclination angle during machining
- Cycle **450 SAVE KINEMATICS** (ISO: **G450**, option 48) automatically creates a log file.

Miscellaneous functions

When tilted machining (option 8), **M128**, or **FUNCTION TCPM** (option 9) is active, the control saves the current positions of the rotary axes to the Q parameters **Q120** to **Q122**.

If you use **M138** to select individual rotary axes for these functions, then the control enters a value of 0 for the rotary axes that have not been selected.

DXF Converter (option 42)

DXF Converter outputs the **FUNCTION MODE MILL** machining mode as a comment.

Program run

- A block scan takes into account a feed rate defined with **M116** (option 8) in mm/min for rotary axes.
- The setting of the feed-rate potentiometer is effective only for the programmed feed rate. As long as the programmed feed rate has not yet been reached, the feed-rate potentiometer has no effect.

Pallets (option 22)

For pallets, you define a line number of the pallet table for the block scan. Within a block scan, you can define the name of the NC program relative to the active pallet table.

Positioning w/ Manual Data Input operating mode

The function **PLANE RESET** (option 8) always positions the rotary axes to their home position, regardless of the 3-D ROT menu settings for manual operation.

Status displays

- When **M136** is active, the control displays the feed rate in mm/1 or inches/1 in the general status display.
- In **Manual Operation** mode, the general status display shows the contouring feed rate for a movement with multiple axes only if 3-D ROT (option 8) is active for manual mode.

Manual touch probe cycles (option 17)

- In the **Hole measurement** and **Measure a stud** functions, you can select the starting angle for the measurement by soft key. The control provides four soft keys based on steps of 90°.
- The **Probing of circular pattern** function has been integrated in the manual **Circle probing** cycle. This function allows you to determine a center point from multiple circles.
- In manual touch probe cycles, the **END** and **ACTUAL POSITION CAPTURE** keys can be used instead of the corresponding soft keys.
- The manual touch probe cycles adapt the displayed measured values to the active preset.
- By default, the manual touch probe cycles provide the positive direction of the principal axis for selection.

Tables

- The preset table shows the columns **SPA**, **SPB**, and **SPC**, regardless of the active tool axis.
- The maximum input range of the columns **SPA**, **SPB**, and **SPC** of the preset table has been increased from $\pm 360.0000000^\circ$ to $\pm 999.9999999^\circ$.
- The input range of the column **F** of the touch probe table has been increased from 3000 mm/min to 9999 mm/min.
- When you import a tool table from the iTNC 530, the control defines all unknown tool types as **Undefined**.

Tools

- When you edit a tool in the tool table or tool management, the only table line that is locked for other access is the line containing this tool.
If this tool is called in **Program run, single block** or **Program run, full sequence** operating mode, then the control issues an error message.
- The deletion of a tool from the tool table is not possible if it is stored in the pocket table. You can change this setting in the optional machine parameter **deleteLoadedTool** (no. 125301). The machine manufacturer enables this parameter.

MOD functions

- The control checks the values defined in the **Traverse limits** MOD function for their validity. If a value is not valid, the control issues an error message and highlights the input field in red.
- The control indicates for every axis whether the traverse limits are active or inactive.

5

Software 81760x-04

5.1 New functions

Further information: User's Manual for Conversational Programming (ID: 1096883-13)

Further information: User's Manual for Cycle Programming (ID: 1096886-13)

NC fundamentals

The control supports programming of spatial arcs. Spatial arcs are defined based on three axes and are perpendicular to the working plane.

Program functions

FUNCTION DWELL allows you to program a dwell time in seconds or define the number of spindle revolutions for dwelling.

Programming Q parameters

- The **FN 0** function allows you to assign the **Undefined** status to Q parameters.
- The **FN 9** function allows you to compare QS parameters and texts for equality.
- In the mask file of the function **FN 16: F-PRINT (ISO: D16)**, you can define whether the Q-parameter output is right-aligned or left-aligned.
- The mask file for **FN 16: F-PRINT (ISO: D16)** supports the following languages:
 - Chinese
 - Chinese (traditional)
 - Norwegian
 - Romanian
 - Slovak
 - Turkish
- The **FN 18: SYSREAD** functions (ISO: **D18**) have been enhanced:
 - **FN 18: SYSREAD (D18) ID220 NR3**: difference between the reference point and the preset of an axis
 - **FN 18: SYSREAD (D18) ID320 NR3**: current machining time of the active NC program in minutes
 - **FN 18: SYSREAD (D18) ID630 NR2**: SIK ID of the control
 - **FN 18: SYSREAD (D18) ID975 NR1**: tool usage test for the current NC program
 - **FN 18: SYSREAD (D18) ID990 NR10**: tool number for a tool name
- The function **FN 38: SEND (ISO: D38)** is available without having to enter a code number.
- The function **SYSSTR(ID10630 NR10)** allows you to determine the NC software version.

Programming aids

- The screen layout **PROGRAM + SECTS** is available in the operating modes **Program run, single block** and **Program run, full sequence**.
- The control highlights the current structure block in light gray for improved orientation.

Test run (option 20)

- Press the **MEASURING** soft key to have the control display the workpiece coordinates at the position of the mouse pointer in 3-D view and plan view.
- The **FMAX PATHS** soft key allows you to select whether the control shows or hides the rapid-traverse movements of the tool.
- The **RESET TOOL PATHS** soft key allows you to delete the displayed tool paths during simulation.
- The **RESET THE VOLUME MODEL** soft key allows you to reset the workpiece to its blank form during simulation.
- The **STOP AT** soft key allows you to define an NC block at which the simulation should stop.

File management

- The file manager displays vertical scrollbars and allows for quick navigation through scrolling.
- The file manager shows the active type filter in the displayed path.
- In the selection window that appears after pressing the **SAVE AS** soft key, you can open the folder at the current cursor position using the **SWITCH** soft key.
- The selection window that appears after pressing the **LAST FILES** soft key also shows files that have been saved using **SAVE AS**.

Program run

- If program run has been canceled, the control saves the point of interruption (i.e., the last active NC block). The **SELECT LAST** soft key allows you to call this NC block from within the extended block scan function
- If, while restarting, the control finds a saved point of interruption, then it displays a message in **Program run** operating mode, informing you that a block scan to the saved position can be performed.
- The **CONTINUE BLOCK SCAN** soft key allows you to perform a multi-level block scan (e.g., to execute a touch probe cycle).
- The **SELECT** soft key that allows you to select the file paths of NC programs or tables from a pop-up window has been added to the block-scan soft-key row.
- In **RETRACT** operating mode, the control shows the active machine kinematics.
- The **CANCEL THE FEED RATE LIMITATION** soft key allows you to deactivate the feed-rate limit for the **RETRACT** operating mode.

Positioning w/ Manual Data Input operating mode

The **Positioning w/ Manual Data Input** operating mode has been enhanced.

The following functions are supported:

- Execution of connected program sequences (e.g., contours)
- Programming of subprograms and program section repeats
- Programming of radius compensation **RL** (ISO: **G41**) and **RR** (ISO: **G42**)
- Copying, cutting, and pasting of NC blocks and blocks
- Programming of Q parameters
- Programming of program functions (e.g., **FUNCTION TCPM**) (option 9)

When an NC block has been executed without error, the control is in an interrupted state. In this state, the control permits various actions (e.g., cursor movements), but they can lead to the loss of modally effective program information.

Status displays

- When a pulsing spindle speed is active, the control shows a corresponding icon in the general status display.
- When an NC block has been executed without error in the **Positioning w/ Manual Data Input** operating mode, the control displays the **control-in-operation** icon in transparent red in the general status display.
- The **CYC** tab of the additional status display contains the maximum values of the parameters **T** and **TA** from Cycle **32 TOLERANCE** (ISO: **G62**).
- The **Basic transformations** area has been added to the **POS** tab of the additional status display. This area contains the active **SPA, SPB, SPC, X, Y, and Z** transformations.

Moving the machine axes

- You can connect up to five portable handwheels to the control.
- You can use the axis keys of the control to select the axis to be traversed with an HR 130 panel-mounted handwheel.
- The control uses the active unit of measure (mm or inches) to include movements traversed with the handwheel in calculations.
- If you have connected a handwheel with display to the control, you can use the handwheel for positioning within the manual touch probe cycles (option 17).

Tilting the working plane (option 8)

In the 3-D ROT menu, the control shows the active machine kinematics.

Tools

In the **OVRTIME** column of the tool table, you can define a time period for which the tool is permitted to be used beyond the maximum tool life. The machine manufacturer inserts this table column.

Error messages and help system

The control displays the various error classes in the following colors:

- Red for errors
- Yellow for warnings
- Green for notes
- Blue for information

HEROS functions

The **NC/PLC Backup** and **NC/PLC RESTORE** HEROS functions allow you to back up and restore the **TNC** drive or individual folders. The machine manufacturer can also create a complete backup of the machine configuration.

Machine-specific user parameters

- The optional machine parameter **displayFont** (no. 100812) allows you to reduce the font size of the machine operating modes to the size of the **Programming** operating mode.
- In the optional machine parameter **noParaxMode** (no. 105413), you can disable the parallel-axis function **FUNCTION PARAXMODE**.

5.2 Modified functions

Further information: User's Manual for Conversational Programming (ID: 1096883-13)

Further information: User's Manual for Cycle Programming (ID: 1096886-13)

NC fundamentals

- Cylindrical workpiece blanks can also be defined by means of a diameter.
- You can use the transition elements **RND** (ISO: **G24**) and **CHF** (ISO: **G25**) between three-dimensional contours, such as a helix.
- Within the **PATTERN DEF** function, you need to define only the first position in absolute coordinates.
- When simulating or executing the miscellaneous function **M124**, the control no longer aborts with an error message. Instead, it skips the function and displays a warning. This function may be contained in NC programs imported from previous control models.

Machining cycles

- Cycle **225 ENGRAVING** (ISO: **G225**) has been enhanced.
 - The parameter **Q367 TEXT POSITION** allows you to define the starting position based on the coordinates programmed in the cycle call.
 - The parameter **Q574 TEXT LENGTH** allows you to define the maximum text length. If a value of 0 is defined in parameter **Q513 CHARACTER HEIGHT**, the control adapts the character height to the maximum text length.
 - Input option 2 has been added to the parameter **Q516 TEXT ARRANGEMENT**. If you select this option, the text is engraved along the inside of a circular arc.
- The minimum input value of parameter **Q370 MAX. OVERLAP** in Cycle **232 FACE MILLING** (ISO: **G232**, option 19) has been changed from 0.1 to 0.001.
- Path generation has been optimized in Cycle **233 FACE MILLING** (ISO: **G233**, option 19). The number of lateral infeeds has been reduced.
- The parameter **Q439 FEED RATE REFERENCE** has been added to Cycle **251 RECTANGULAR POCKET** (ISO: **G251**, option 19). This parameter allows you to define whether the programmed feed rate is referenced to the cutting edge of the tool or the path of the tool center.
- The control moves the tool to the edge of the pocket on a radial path during the finishing operation of Cycles **251 RECTANGULAR POCKET** (ISO: **G251**) and **252 CIRCULAR POCKET** (ISO: **G252**, option 19).
- The parameter **Q438 ROUGH-OUT TOOL** has been added to Cycle **24 SIDE FINISHING** (ISO: **G124**, option 19). This parameter allows you define the tool for roughing out the contour.
- Cycle **275 TROCHOIDAL SLOT** (ISO: **G275**, option 19) has been enhanced:
 - The parameter **Q369 ALLOWANCE FOR FLOOR** allows you to define a finishing allowance for the floor.
 - The parameter **Q439 FEED RATE REFERENCE** allows you to define whether the programmed feed rate is referenced to the cutting edge of the tool or the path of the tool center.

Touch probe cycles (option 17)

- Cycles **460 CALIBRATION OF TS ON A SPHERE** (ISO: **G460**), **462 CALIBRATION OF A TS IN A RING** (ISO: **G462**), and **463 TS CALIBRATION ON STUD** (ISO: **G463**) save the measured position of the circle center to the Q parameters **Q151** and **Q152**.
- Input option 2 has been added to the parameter **Q340 CHECK** in Cycles **481 CAL. TOOL LENGTH** (ISO: **G481**), **482 CAL. TOOL RADIUS** (ISO: **G482**), and **483 MEASURE TOOL** (ISO: **G483**). If this input option is selected, the control compares the measured length or radius with the values in the tool table and saves the difference to the Q parameters **Q115** and **Q116**. Tool compensation is not automatically performed by the control.
- The logs of Cycles **451 MEASURE KINEMATICS** (ISO: **G451**) and **452 PRESET COMPENSATION** (ISO: **G452**, option 48) contain the positions of the tested rotary axes before and after optimization.

Program functions

FUNCTION TCPM (option 9) with selected **AXIS SPAT** takes into account an active mirror function that has been programmed with Cycle **8 MIRRORING** or an active rotation that has been programmed with Cycle **10 ROTATION**.

Programming Q parameters

If you determine the length of an undefined Q parameter by means of the function **STRLEN**, the control displays a value of -1.

Calling an external NC program

If an NC program called with **CALL PGM** (ISO: %) includes an **M2** or **M30** function, then the control displays a warning.

Programming aids

The colors for the programming graphics have been changed:

- Blue for contour elements that have completely been defined
- Violet for contour elements that have not yet been completely defined (e.g., chamfers or FK sequences)
- Ocher for paths of the tool center
- Red for rapid-traverse movements

Test run (option 20)

- If you select a new NC program or start it with the **RESET + START** soft key, the control resets the tool data.
- In **BLANK IN WORK SPACE** view, the control displays the datum of the worktable as the reference point. The **MACHINE REF POINT** soft key allows you to select whether the control shows or hides the reference point.

File management

- When a large file is transferred to a USB device, the control displays the warning **Write access to USB device** during file transfer.
- The **RENAME** soft key allows you to change upper and lower cases of a file name.
- The positions of the soft keys provided for the **Copy: Select the target directory** function have been modified.
- The **SHOW ALL** soft key provided in the soft-key row for the **SELECT TYPE** function is available in all operating modes.

DXF Converter (option 42)

- A hyphen is permitted in the name of a file saved from the DXF Converter.
- The dialog window for saving files has been modified to match the HEROS dialogs. The dialog window texts are displayed in the active language.

Program run

- If you modify the preset during a program run interruption, you can resume the NC program run only with a block scan or **GOTO**.
- Entering into an **FK** sequence (option 19) by means of a block scan works properly.
- The control issues an error message when a touch probe cycle (option 17) is run within the NC program without a touch probe being inserted.

Pallets (option 22)

While a pallet table is being run, you can edit it only by using the **EDIT PALLET** soft key.

Status displays

- On the **Overview** tab of the additional status display, the control displays no more than 32 characters in the **LBL** field.
- The control displays the path of the active NC program on the **Overview** and **PGM** tabs of the additional status display.
- The control displays the tool measurement values on the **TT** tab of the additional status display even if you select the tab during or after measurement.
- The **Next tab** key allows you to toggle between the tabs of the additional status display.

Moving the machine axes

- The positions of the soft keys provided for the **Jog increment** function have been modified.
- A feed rate limit set with the **F MAX** soft key is effective in all operating modes.

Manual touch probe cycles (option 17)

- The manual touch probe cycle **Corner as preset** allows you to probe both a circular stud and a hole on a straight line.
- The positions of the soft keys provided in the evaluation menu have been standardized for all manual touch probe cycles.
- If you use a manual touch probe cycle to activate a preset, then the existing transformations are retained (e.g., a datum shift).

Tilting the working plane (option 8)

If you activate the **Tilt working plane** function in the 3-D ROT menu, then the control uses the **COORD ROT** transformation type by default.

Tables

- When you open the preset table, the cursor is on the line of the active preset.
- When you import a tool table from the iTNC 530, the control correctly loads the numerical values from the **R-OFFS** column.

Tools

- You are allowed to use the special characters **%** and **,** in tool names.
- When you create a new tool, the columns **L** and **R** of the tool table are empty and therefore not defined. The tool cannot be inserted until these columns have been defined.
- The default value of the **LIFTOFF** column of the tool table has been changed from **Y** to **N**.
- The **SELECT** soft key allows you to define a replacement tool in the **RT** column of the tool table.

MOD functions

The settings for a DHCP server have been modified. The control reserves an IP address for the time period defined in the **Lease Time (hours):** dialog following a power interruption.

HEROS functions

The HEROS menu has been restructured:

- The **HePacketManager** function is now in **Settings**.
- The **perf2** function is now in **Diagnostic**.
- The function for taking **screenshots** is now in **Tools**.

6

Software 81760x-05

6.1 New software options

Further information: User's Manual for Conversational Programming (ID: 1096883-15)

Batch Process Manager (option 154)

This option enables job planning at the machine. **Batch Process Manager** runs on the control's third desktop.

You create your job lists, including pallets, setups, and NC programs, and run them using the pallet management (option 22).

Batch Process Manager shows the following information regarding the job lists:

- Correctness of NC program
- Run time of NC programs
- Availability of tools
- Times at which manual interventions at the machine are necessary

Renamed software options

Option 42 was renamed as **CAD Import**.

6.2 New functions

Further information: User's Manual for Conversational Programming (ID: 1096883-15)

Further information: User's Manual for Cycle Programming (ID: 1096886-14)

NC fundamentals

The control supports up to eight control loops, of which up to two are configurable as spindles.

Machining cycles

Cycle **276 THREE-D CONT. TRAIN** (ISO: **G276**, option 19)

This cycle allows you to machine a contour train of different depths. You define the contour in a subprogram and select it with Cycle **14 CONTOUR** (ISO: **G37**). Using Cycle **270 CONTOUR TRAIN DATA** (ISO: **G270**), you add the approach motion, departure motion, and radius compensation.

Touch probe cycles (option 17)

Cycle **441 FAST PROBING** (ISO: **G441**)

This cycle allows you to define various parameters (e.g., the positioning feed rate) for all touch probe cycles that are subsequently used in the NC program.

Program functions

- **FUNCTION COUNT** allows you to define a counter (e.g., for counting the number of produced workpieces) in the NC program. The control considers the counter in the operating modes **Program run, single block** and **Program run, full sequence**.

FUNCTION COUNT includes the following subfunctions:

- Resetting of counter value
- Definition of target value of the counter
- Setting of counter to a desired value
- Incrementing of counter value
- Definition of automatic program section repeat if target value has not been reached
- **FUNCTION LIFTOFF** retracts the tool from the contour upon an NC stop. You define the direction of the movement in the NC program. The value **Y** must be defined in the **LIFTOFF** column of the tool table.
- When using **FUNCTION PROG PATH** (option 9), the control also applies the 3-D radius compensation to the active tool radius **R + DR** or the corner radius **R2 + DR2**, and no longer only to the delta value **DR**.
- The **CONTOUR DEF** function also allows you to define a simple contour formula in ISO programming mode.
- When programming **PLANE** functions (option 8), you can also use **FMAX** and **FAUTO** in ISO programming mode.

Programming Q parameters

- In the function **FN 16: F-PRINT** (ISO: **D16**), you can use a Q or QS parameter to define the path of the source file and the output file. You first need to define the parameter in the NC program.
- The **FN 18: SYSREAD** functions (ISO: **D18**) have been enhanced:
 - **FN 18: SYSREAD (D18) ID10**
 - **NR6**: number of the touch probe cycle that was last executed
 - **NR7**: type of program that calls the macro (e.g., NC program, cycle)
 - **FN 18: SYSREAD (D18) ID220 NR4**: value for the OEM offset of an axis
 - **FN 18: SYSREAD (D18) ID280 NR1**: check for whether **M128** is active
 - **FN 18: SYSREAD (D18) ID295**
 - **NR4**: check for whether a rotary axis is included in the kinematic calculation
 - **NR10**: axis ID for an axis index
 - **NR11**: axis index for an axis ID
 - **FN 18: SYSREAD (D18) ID510**
 - **NR5**: check for whether a clearance height is programmed for an axis during tool-oriented pallet machining
 - **NR6**: value for clearance height
 - **FN 18: SYSREAD (D18) ID610**: values of various machine parameters for **M120**
 - **FN 18: SYSREAD (D18) ID630 NR10**: iTNC 530 or NCK-based control model
 - **FN 18: SYSREAD (D18) ID920**
 - **NR1**: target value of the counter
 - **NR2**: current value of the counter
 - **NR12**: difference between the current value and the target value
 - **FN 18: SYSREAD (D18) ID992**
 - **NR17**: check for whether the programming graphics are active
 - **NR18**: check for whether the **AUTO DRAW** soft key is active
- The **D00** function allows you to assign the status **Undefined** to Q parameters in ISO programming mode.
- The **D09** function allows you to define an NC program jump that is linked to a condition.

In ISO programming mode, the following functions are available in addition to the comparison with a value:

- **IS DEFINED** causes the jump to be made at a defined Q parameter.
- **IS UNDEFINED** causes the jump to be made at an undefined Q parameter.

Calling an external NC program

The **SELECT CYCLE** soft key has been added to the soft-key row for the **PGM CALL** functions. This soft key activates the function **SEL CYCLE** (DIN/ISO: **G: :**) that allows you to use any NC program as a machining cycle.

Programming aids

- You can turn an NC block into a comment by pressing the **INSERT COMMENT** soft key or the < key of the alphabetic keyboard.
- You can turn a comment into an active NC block by pressing the **REMOVE COMMENT** soft key or the > key of the alphabetic keyboard.

Test run (option 20)

Using the **SELECT TRAVERSE RANGE** soft key from within the **BLANK IN WORK SPACE** menu, you can select ranges of traverse defined by the machine manufacturer for the **Test Run** operating mode.

File management

- You can expand and collapse elements in all tree structures by double-clicking them.
- The additional **ToolHolderWizard** tool allows you to open tool carrier files from within the file manager.

CAD Viewer

- The **CAD-Viewer** can open STEP (*.stp and *.step) and IGES (*.igs and *.iges) files. **CAD Import** (option 42) additionally allows you to extract contours or machining positions from these files.
- When more than one file is opened in the **CAD-Viewer**, then the control reduces the size of the additional **CAD-Viewer** windows shown on the third desktop.

Pallets (option 22)

- Tool-oriented pallet machining allows you to machine several workpieces in one setup without the need for a pallet changer. For this purpose, the machine manufacturer expands the pallet table.

You define the following information for tool-oriented pallet machining:

- Workpiece status in the **W-STATUS** column
- Machining method in the **METHOD** column
- Optional definition of a clearance height for each axis
- The pallet preset table allows you to compensate for mechanical deviations between individual pallets, for example.
- In the pallet table, you can insert the **PALPRES** column for defining the pallet preset.
- The control displays the current pallet preset on the **PAL** tab of the additional status display.

Status displays

When mirroring is active, the control displays a corresponding icon in the general status display.

Manual touch probe cycles (option 17)

- In the manual touch probe cycle **Corner as preset**, you can compensate for angular misalignment by rotating the rotary table.
- The **TCH PROBE MONITOR OFF** soft key allows you to suppress touch probe monitoring for 30 seconds. This means that retraction is possible when the stylus is deflected, for example.
- If the function for orienting the spindle to the probing direction is active, the number of spindle revolutions is limited when the guard door is open. If, for example, a number of revolutions that is greater than the maximum permissible number is required for touch probe cycles, then the direction of spindle rotation changes and the traverse path therefore also changes.

Tables

- You can manually edit the line 0 of the preset table.
- The machine manufacturer can disable the setting of presets for individual axes.
- The **SERIAL** column has been added to the touch probe table (option 17). For touch probes with EnDat interface, the control automatically enters the serial number into this column.
- The functions for importing and modifying tables have been merged into the **ADAPT NC PGM / TABLE** function. You can also import or modify freely definable tables. The control recognizes whether a table is imported or the table format is adapted.
- The machine manufacturer can define update rules for importing tables (e.g., for the automatic removal of German umlauts from tables and NC programs).

Tools

- You can search for a tool name in the tool table by entering the name using the alphabetic keyboard. After entering the first character, the control opens a field containing the search criterion and highlights the first tool whose name matches it. If you re-enter the most recently entered character, then the control highlights the next search result.
- The tool management supports touch probes with EnDat interface.

Accessories

The control supports operation of a touchscreen. The touchscreen can be operated by gestures as well as by keys.

MOD functions

- The **Set up touch probes** function for managing different touch probes and transceiver units has been added to the **Machine settings** group. The control opens the dialog on its third desktop.

HEROS functions

- The **Touchscreen Calibration** function allows you to increase the accuracy of the screen.
- The **Touchscreen Configuration** function allows you to define the touchscreen properties (e.g., the sensitivity).
- The **Touchscreen Cleaning** function allows you to lock the screen for cleaning.

Machine-specific user parameters

- The machine parameter **decimalCharakter** (no. 100805) allows you to select whether the control uses a period or a comma as the decimal separator.
- The machine parameter **iconPrioList** (no. 100813) allows you to prioritize the icons of the general status display. The control displays the icons in the defined sequence.
- The optional machine parameter **suppressResMatlWar** (no. 201010) allows you to define whether the control suppresses the **Residual material** warning.
- The optional machine parameter **clearPathAtBlk** (no. 124203) allows you to define whether, with a new **BLK FORM** (ISO: **G30/G31**), the control clears the tool paths in **Test Run** operating mode.
- The optional machine parameter **transDatumCoordSys** (no. 127501) allows you to define whether the control displays a datum shift in the coordinate system of the workpiece or in the coordinate system of the tilted plane. The machine manufacturer enables this parameter.

6.3 Modified functions

Further information: User's Manual for Conversational Programming (ID: 1096883-15)

Further information: User's Manual for Cycle Programming (ID: 1096886-14)

NC fundamentals

- You can change the sorting sequence within the pop-up window for tool selection by clicking the header of a column. The sorting sequence and the width of the individual columns are retained across power cycles of the control.
- The coordinates for defining a vector in **LN** blocks are evaluated very accurately, regardless of option 23. You can define a maximum of seven decimal places.
- Pasting a large amount of data into an NC program has become faster.

Machining cycles

- Cycle **225 ENGRAVING** (ISO: **G225**) allows you to engrave the current count.
- Cycles **256 RECTANGULAR STUD** (ISO: **G256**) and **257 CIRCULAR STUD** (ISO: **G257**, option 19) now also provide roughing and finishing operations.
The following parameters have been added:
 - **Q215 MACHINING OPERATION**
 - **Q369 ALLOWANCE FOR FLOOR**
 - **Q338 INFEED FOR FINISHING**
 - **Q385 FINISHING FEED RATE**
- During finishing in Cycle **233 FACE MILLING** (ISO: **G233**, option 19), the control monitors the length of the tool's cutting edge (**LCUTS**) and reduces the infeed depth **Q338** where needed.
- If, in Cycle **233 FACE MILLING** (ISO: **G233**, option 19), you enter a value greater than 1 for the parameter **Q370 TOOL PATH OVERLAP**, then the cycle also takes this value into account during the first cut.
- Cycle **25 CONTOUR TRAIN** (ISO: **G125**, option 19) has been enhanced to enable the machining of residual material.
The following parameters have been added:
 - **Q18 COARSE ROUGHING TOOL**
 - **Q446 RESIDUAL MATERIAL**: accepted residual material thickness on the finished part
 - **Q447 CONNECTION DISTANCE**: maximum distance between two areas of residual material, between which the control does not lift off the tool
 - **Q448 PATH EXTENSION**: approach path and departure path for every removal of residual material
- If an approach block or a departure block leads to contour damage in Cycles **25 CONTOUR TRAIN** (ISO: **G125**) or **276 THREE-D CONT. TRAIN** (ISO: **G276**, option 19), then the control interrupts the machining process and issues an error message.
- The control supports the use of the **TRANS DATUM AXIS** function in the contour description of SL cycles (option 19).
- In Cycles **205 UNIVERSAL PECKING** (ISO: **G205**) and **241 SINGLE-LIP D.H.DRLNG** (ISO: **G241**, option 19), the plunging behavior has been modified in order to prevent the tool from becoming damaged.
If you have defined a deepened starting point in the parameter **Q379 STARTING POINT**, then the control activates the spindle speed and the feed rate at a calculated position above the deepened starting point.
- Cycle **239 ASCERTAIN THE LOAD** (ISO: **G239**, option 143) can adapt the maximum acceleration of the axes depending on the weight, and ascertain the load on synchronized axes (gantry axes) if they share one common position encoder. Synchronized

axes are programmed using one common axis and move synchronously.

- The **OLD CYCL** soft key and the associated cycles have been removed:
 - Cycle **1 PECKING** (ISO: **G83**)
 - Cycle **2 TAPPING** (ISO: **G84**)
 - Cycle **3 SLOT MILLING** (ISO: **G74**)
 - Cycle **4 POCKET MILLING** (ISO: **G75/G76**)
 - Cycle **5 CIRCULAR POCKET** (ISO: **G77/G78**)
 - Cycle **17 RIGID TAPPING** (ISO: **G85**)
 - Cycle **210 SLOT RECIP. PLNG** (ISO: **G210**)
 - Cycle **211 CIRCULAR SLOT** (ISO: **G211**)
 - Cycle **212 POCKET FINISHING** (ISO: **G212**)
 - Cycle **213 STUD FINISHING** (ISO: **G213**)
 - Cycle **214 C. POCKET FINISHING** (ISO: **G214**)
 - Cycle **215 C. STUD FINISHING** (ISO: **G215**)
 - Cycle **230 MULTIPASS MILLING** (ISO: **G230**)
 - Cycle **231 RULED SURFACE** (ISO: **G231**)

You can no longer insert these cycles in an NC program, but you can still edit and run them.

Touch probe cycles (option 17)

The machine manufacturer can define whether the offset **R-OFFS** of the tool table is taken into account for a tool without cutting edges. If the offset is taken into account, measurement with a tool touch probe is performed with a rotating spindle.

Program functions

- In **FUNCTION TCPM** (option 9), you can define the tool reference point and the center of rotation. The control positions the tool to the tool reference point during machining and rotates the tilting axis around the center of rotation during tilting.

The following options are available:

- **REFPNT TIP-TIP** (default): the tool reference point and the center of rotation are located at the tool tip
 - **REFPNT TIP-CENTER**: the tool reference point is located at the tool tip, and the center of rotation at the center of the tool-tip radius. This is suitable only for turning tools .
 - **REFPNT CENTER-CENTER**: the tool reference point and the center of rotation are located at the center of the tool-tip radius.
- If the file to be deleted using **FUNCTION FILE DELETE** does not exist, the control does not issue an error message.

Miscellaneous functions

- With **M94**, the control reduces the display value of all rotary axes to a value of less than 360°, even if the axes are not modulo axes. **M94** does not become effective if a rotary axis is limited by software limit switches or traverse limits.
- **M138** allows you to select the rotary axes that the control takes into account in tilted machining (option 8), **M128**, or **FUNCTION TCPM** (option 9).

The machine manufacturer defines whether, for the rotary axes that are not selected, the control enters a value of 0 into the Q parameters **Q120** to **Q122** or takes into account the current axis angle.

Programming Q parameters

The **SYSSTR(ID10010 NR1)** function allows you to read the file path of a pallet program.

Programming aids

- The programming graphics displays holes and threads in light blue.
- When a locked tool is called in an NC program, the control displays a warning instead of an error message during simulation in the programming graphics. When the **AUTO DRAW** soft key is active, the control does not issue a message.

Test run (option 20)

- The machine manufacturer can change the tool change position for **Test Run** operating mode.
- The control displays the tool in blue during an air cut, and in red during workpiece contact.
- In the view in three planes, the positions of the sectional planes are retained when a new program or a new **BLK FORM** (ISO: **G30/G31**) is selected.
- The control saves graphic settings across power cycles of the control (e.g., workpiece shown in color or shades of gray).
- When a locked tool is called in an NC program, the control displays a warning instead of an error message.
- When configuration subfiles are loaded during a test run, the control displays a warning instead of an error message.

File management

The control automatically detects a connected USB device. The soft key for connecting a USB device has been removed.

Program run

- When returning to the contour, the last axis positioned by the control is, by default, the tool axis. If the tool tip is positioned below the point of approach before returning to the contour, then the first axis traversed by the control is the tool axis.
- If **M101** is active, the control inserts a replacement tool upon expiration of the tool life. When returning to the contour after the tool change, the last axis positioned by the control is, by default, the tool axis. If the tool tip is positioned below the point of approach before returning to the contour, then the first axis traversed by the control is the tool axis. If an active retraction with **M140** is programmed, this positioning logic is not taken into account.

Status displays

- The control displays the spindle speed with at least four digits in the general status display. This means if the spindle speed is less than 1000, then the decimal places are also displayed.
- The icons for the basic rotation, 3-D basic rotation, and tilted working plane (option 8) have been modified to make them easier to distinguish.
- The **FUNCTION TCPM** (option 9) icon in the general status display has been modified.

Moving the machine axes

- In a tilted working plane (option 8), you can also home axes (e.g., a tool magazine) that are not activated in the active kinematic model.
- In **Manual Operation** mode, you can define the spindle speed with up to three decimal places.
- On handwheels with display, the speed for setting the jog increment, spindle speed, and feed rate has been adapted for increased ease of use.
- The control issues a warning if the handwheel potentiometers are still active when a handwheel is deactivated.
- If the battery voltage of the HR 550 or HR 550FS electronic handwheel is too low, the control issues a warning.

Tables

- You can neither set nor modify a preset in the preset table without having homed the axes.
- In some table fields, you can select the values from a pop-up window. You can open the pop-up window by pressing the **ENT** key or by double-clicking.

Tools

When you place the cursor into an input field in the form view of the tool management, the control selects the complete field contents. If you enter a new value, then the complete contents are overwritten.

Error messages and help system

- When displaying error messages, the control sorts them first by priority and, second (within the same priority), by time. Short-term informational texts are always displayed, regardless of the priority.
- If an NC program called with **CALL PGM** (ISO: %) includes an **M2** or **M30** function, then the control displays a warning. If you then select another NC program, the control clears the warning.

Remote Desktop Manager (option 133)

In the **Remote Desktop Manager**, all of the functions that you previously had to call by clicking the right mouse button have been added to the menu bar

The following functions have been added to the menu bar:

- **Start the connection**
- **Terminate the connection**
- **Import connections**
- **Export the connections**

7

Software 81760x-06

7.1 New software options

Further information: User's Manual for Setup, Testing and Running NC Programs (ID: 1263172-10)

Further information: User's Manual for Cycle Programming (ID: 1096886-15)

State Reporting Interface (option 137)

SRI offers an interface for acquiring information about the operating states and status changes of your machine.

SRI stores historical production data and makes them available via an HTML log. The operating data can still be retrieved even after the corporate network has been down for hours at a time.

Monitoring of configured machine components for overload (option 155)

Component monitoring allows the machine manufacturer to define load limits and overload reactions for individual machine components.

The **CM** and **CM Detail** tabs that display the components being monitored and their current values have been added to the additional status display.

The machine parameter **CfgComoUserData** (no. 129400) allows you to select whether the control performs the reactions defined for errors and warnings. The machine manufacturer enables this parameter.

7.2 New functions

Further information: User's Manual for Conversational Programming (ID: 1096883-16)

Further information: User's Manual for Setup, Testing and Running NC Programs (ID: 1263172-10)

Further information: User's Manual for Cycle Programming (ID: 1096886-15)

NC fundamentals

- You can use the unlabeled key between **PGM MGT** and **ERR** to toggle between the operating modes.
- In the **TOOL DEF** (ISO: **G51**) function, you can define the tool name using a QS parameter.
- The **PLANE XY ZX YZ** soft key allows you to change the working plane for **FK** programming (option 19), regardless of the programmed **BLK FORM** (ISO: **G30/G31**).
- Without the **END PGM** NC block (ISO: **G71**), an NC program is incomplete. When you open an incomplete NC program in **Programming** operating mode, the control issues a warning and automatically adds the missing NC block. When you start an incomplete NC program in **Program run, single block** or **Program run, full sequence** operating mode, the control issues an error message.

Touch probe cycles (option 17)

The following cycles allow you to ascertain the exact workpiece position and compensate it as needed during program run. The cycles save the probing positions and the ascertained values to Q parameters and create a log.

- Cycle **1410 PROBING ON EDGE** (ISO: **G1410**)
This cycle allows you to ascertain the angle of an edge relative to the principal axis as well as the positions of the touch points.
- Cycle **1411 PROBING TWO CIRCLES** (ISO: **G1411**)
This cycle allows you to ascertain the center points of two holes or studs and the angle of the line connecting both center points.
- Cycle **1420 PROBING IN PLANE** (ISO: **G1420**)
This cycle allows you to ascertain the angle of a plane and the positions of the touch points.

Program functions

The **PLANE** functions (option 8) provide the alternative selection option **SYM** in addition to **SEQ**. If **SYM** is selected, the control calculates the possible solutions for tilting based on the symmetry point of the axis, and not on the home position. This means that **SYM** does not require a pre-positioning movement for any of the possible solutions.

Programming Q parameters

- The following options have been added to the text file that defines the output format for **FN 16: F-PRINT** (ISO: **D16**):
 - The character ***** allows you to create a comment line in the text file. The control does not show comment lines in the log.
 - The output format **%RS** allows you to define that the control outputs the text of a QS parameter to the log without formatting.
- The **FN 18: SYSREAD** functions (ISO: **D18**) have been enhanced:
 - **FN 18: SYSREAD (D18) ID90 NR2**: maximum speed of a spindle (**IDX**; e.g., **IDX99** for active spindle)
 - **FN 18: SYSREAD (D18) ID280 NR3**: current status of **FUNCTION TCPM**
- The functions **FN 27: TABWRITE** (ISO: **D27**) and **FN 28: TABREAD** (ISO: **D28**) support QS parameters that allow you to write to and read from text fields of a freely definable table.

Programming aids

- The cutting data calculator enables you to use cutting data tables for automatically calculating spindle speeds and feed rates.
You create the following tables for this purpose:
 - Workpiece material table (WMAT.tab)
 - Tool material table (TMAT.tab)
 - Cutting data table (*.cut or *.cutd)
 You define the tool material, as well as the cutting data table used, in the tool table on a tool-specific basis.
- The cutting data calculator has been modified to allow the use of cutting data tables. You can select the tool, activate the cutting data table, and define the workpiece material. To load the calculated value, press the **APPLY** soft key.

Test run (option 20)

- The control takes into account a programmed counter in **Test Run** operating mode.
- For **Test Run** operating mode, the machine manufacturer can define whether the **TOOL TABLE** soft key opens the tool table or tool management.

File management

- The control supports USB devices with the exFAT file system.
- The additional **Web Browser** tool automatically checks whether updates are available. To update the browser, you temporarily need to deactivate the SELinux security software.
- For touchscreens, the control automatically terminates full-screen mode after five seconds.

CAD Viewer

In the **CAD-Viewer**, you can define the preset and datum by entering the values into the list view window.

Batch Process Manager (option 154)

- **Batch Process Manager** has been integrated in the control as a selectable editor for pallet tables.
- In **Program run, single block** and **Program run, full sequence** operating modes, you can select the **Batch Process Manager** as screen layout. All of the functions available for editing the pallet table are provided only in **Programming** operating mode.

Program run

You can edit a called NC program as soon as program run has been completed.

Positioning w/ Manual Data Input operating mode

In **Positioning w/ Manual Data Input** operating mode, the **HIDE** soft key that enables you to skip NC blocks has been added.

Status displays

- In the general status display, the control displays a feed rate value of less than 10 with one decimal place and a feed rate value of less than 1 with two decimal places.
- If the active preset contains a basic rotation (option 8), the control displays a corresponding icon in the general status display.
- If the active preset contains a 3-D basic rotation (option 8), the control displays a corresponding icon in the general status display.
- The maximum values of **T** and **TA** have been removed from the **CYC** tab of the additional status display.
The control shows the active limits for path tolerance and angle tolerance from one of the following sources:
 - Values from Cycle **32 TOLERANCE** (ISO: **G62**)
 - Values defined by the machine manufacturer

Manual touch probe cycles (option 17)

- The manual probing functions take an active **M128** or **FUNCTION TCPM** (option 9) into account.
- When using the manual touch probe cycle **Probe in plane** for aligning a 3-D basic rotation via the rotary axes (option 8), the control provides the following input options:
 - Feed rate for tilting
 - **TURN** or **MOVE** positioning behavior
 - **NO SYM**, **SYM +**, or **SYM -** selection option

Tilting the working plane (option 8)

The appearance of the 3-D ROT menu has been revised. You can individually activate the settings for **Manual operation** mode and take into account a basic rotation.

Tables

The **REACTION** column for defining the response of the control when a collision of the touch probe occurs has been added to the touch probe table (option 17).

Tools

- You define the tool material in the **TMAT** column of the tool table.
- In the **CUTDATA** column, you select the cutting data table for the cutting data calculator.
- The following tool types have been added:
 - **Ball-nose cutter, BALL**
 - **Toroid cutter, TORUS**
- The following tool types have been removed:
 - **Rough/finish cutter, MILL_RF**
 - **Floor finisher, MILL_FD**
 - **Side finisher, MILL_FS**
 - **Face milling cutter, MILL_FACE**

MOD functions

The **HOST COMPUTER MODE** soft key has been added to the MOD function **External access** in the **Machine settings** group. This soft key allows you to connect an external host computer to the control for data transmission purposes, for example. The machine manufacturer enables this function.

HEROS functions

The user administration feature lets you define every machine operator as a separate user with specific access rights.

The user administration is contained on the **Settings** tab of the HEROS menu. To activate the user administration, create the **useradmin** user and set up a database. Then you create individual users and assign roles to them.

A private **HOME:** directory, to which you can save your private NC programs and files, is available to every user.

Machine-specific user parameters

- The machine parameter **CfgProgramCheck** (no. 129800) allows you to define for NC programs and pallet machining whether and when the control creates a tool usage file.
- The machine parameter **CfgThreadSpindle** (no. 113600) is enabled by entering the code number **123**. This parameter contains the following spindle settings for machining a thread:
 - **sourceOverride** (no. 113603): select the active potentiometer during tapping
 - **thrdWaitingTime** (no. 113601): dwell time in seconds on the root of the thread
 - **thrdPreSwitch** (no. 113602): spindle stops and dwells for this time in seconds before reaching the root of the thread
 - **limitSpindleSpeed** (no. 113604): spindle speed limit for Cycles **17 RIGID TAPPING (ISO: G85)**, **18 THREAD CUTTING (ISO: G86)**, and **207 RIGID TAPPING (ISO: G207)**

7.3 Modified functions

Further information: User's Manual for Conversational Programming (ID: 1096883-16)

Further information: User's Manual for Setup, Testing and Running NC Programs (ID: 1263172-10)

Further information: User's Manual for Cycle Programming (ID: 1096886-15)

NC fundamentals

- If, in a **TOOL CALL** block (ISO: **T**), you do not program a tool, but rather the same tool axis of the active tool, then the control no longer runs a tool-change macro. Instead, the control applies the values defined in the NC block to the active tool (e.g., the cutting values or an oversize).
- If you program **M89** in conjunction with **FK** blocks (option 19), then the control issues an error message.

Machining cycles

- The parameter **Q367 SURFACE POSITION** has been added to Cycle **233 FACE MILLING** (ISO: **G233**, option 19). This parameter allows you to define the position of the surface based on the coordinates programmed in the cycle call.
- For the last infeed, Cycle **24 SIDE FINISHING** (ISO: **G124**, option 19) performs the approach motion and departure motion along a tangential helix, in order to avoid surface blemishes on the floor.

Touch probe cycles (option 17)

- Cycles **408 SLOT CENTER PRESET** (ISO: **G408**) to **419 PRESET IN ONE AXIS** (ISO: **G419**) take into account the machine parameter **chkTiltingAxes** (no. 204601) during probing. Depending on the configuration of the machine parameter, the control either does not check for whether the axis angle matches the tilt angle or it issues an error message if the angles do not match.
- Cycle **420 MEASURE ANGLE** (ISO: **G420**) uses the sum of the safety clearances from the touch probe table and the cycle during repositioning.
- The parameters **Q303 MEAS. VALUE TRANSFER** and **Q305 NUMBER IN TABLE** define how the control saves the preset. If **Q303 = 1** and **Q305 = 0**, then the control copies the active preset to line 0 of the preset table and activates the line 0.
- When restoring a kinematic model, Cycle **450 SAVE KINEMATICS** (ISO: **G450**, option 48) changes only those values that differ from the active values.
- If, in Cycles **451 MEASURE KINEMATICS** (ISO: **G451**) and **453 KINEMATICS GRID** (ISO: **G453**, option 48), you set the parameter **Q431 PRESET** to 1, then the control measures the calibration sphere twice. The first measurement enables the control to precisely determine the position of the calibration sphere. The control monitors the programmed radius of the calibration sphere only during the second measurement.

Program functions

FUNCTION TCPM (option 9) supports peripheral milling (this means that it supports 3-D radius compensation with a constant position of the tool tip and active tool radius compensation).

Programming Q parameters

- The functions **M_CLOSE** and **M_TRUNCATE** for **FN 16: F-PRINT** (ISO: **D16**) have the same effect as far as output to the screen is concerned. Both functions overwrite the previous contents of the pop-up window.
- When using the function **SQL UPDATE** or **SQL INSERT** to write a QS parameter to a table, the control checks the length of the string. If the string is longer than the table column, the control issues an error message.

Test run (option 20)

- The additional status display has been standardized in the operating modes **Test Run** and **Manual Operation**.
- The function of the **GOTO** is identical in all operating modes.

CAD Viewer

- **CAD Import** (option 42) outputs 2-D contours by default.
- **CAD Import** (option 42) outputs angles with **PLANE SPATIAL** instead of **PLANE VECTOR**. You can edit these angles in the NC program.

Moving the machine axes

The machine manufacturer can disable individual M functions for **Manual Operation** mode.

Manual touch probe cycles (option 17)

If the axis angle does not match the tilt angle during manual probing, then the control no longer issues an error message. Depending on the configuration of machine parameter **chkTiltingAxes** (no. 204601), the control either does not check the angles or opens the **Working plane is inconsistent** menu.

Tables

- If the active line of the preset table is reactivated, the control reads out the values again.
- The machine manufacturer defines which file types the control is able to import when using the **ADAPT NC PGM / TABLE** function, as well as the scope of the adaptations.

Tools

The machine manufacturer can define default values for the **L-OFFS** and **R-OFFS** columns of the tool table.

Remote Desktop Manager (option 133)

The setting **Additional waiting time:** has been added to the **Switch-off/restart of a computer** function for shutting down an external computer.

If you define the **Max. waiting time (seconds):**, then, while shutting down the computer, the control waits until the port 445 of the computer has been closed so that the computer can no longer be accessed.

Some Windows applications can delay the shutdown of the computer. You can define an additional waiting time to prevent this delay from causing a loss of data or damaging the computer. The additional waiting time begins when the control can no longer access the the computer.

HEROS functions

The screensaver glideshow has been removed.

8

Software 81760x-07

8.1 New software options

Further information: User's Manual for Setup, Testing and Running NC Programs (ID: 1263172-11)

Further information: User's Manual for Cycle Programming (ID: 1096886-16)

HEIDENHAIN OPC UA NC Server (options 56 to 61)

OPC UA provides a standardized interface for secure data exchange between products independent of the manufacturer. HEIDENHAIN provides the **HEIDENHAIN OPC UA NC Server** for data exchange with the control. These software options allow you to create up to six parallel client connections.

The **Connection Assistant** function has been added to the HEROS menu for setting up the connection. If user administration is active, you link the connections to specific users.

Optimized contour cycles (option 167)

The OCM cycles enable the trochoidal milling of closed or open contour geometries that can be combined from several subcontours. High removal rates can be achieved because the control adheres to the programmed path overlap and thus keeps a constant tool contact angle during roughing.

The option includes the following cycles:

- Cycle **271 OCM CONTOUR DATA** (ISO: **G271**)
- Cycle **272 OCM ROUGHING** (ISO: **G272**)
- Cycle **273 OCM FINISHING FLOOR** (ISO: **G273**)
- Cycle **274 OCM FINISHING SIDE** (ISO: **G274**)

Further information: "Machining cycles", Page 85

Display step

The **Display Step** software option (option 23) is available in the standard control version. The display step of the axes is no longer limited to four decimal places.

The machine parameter **displayPace** (no. 101000) allows you to define the display step for the individual axes. The minimum display step of the axes is 0.01 µm or 0.00001°.

8.2 New functions

Further information: User's Manual for Conversational Programming (ID: 1096883-17)

Further information: User's Manual for Setup, Testing and Running NC Programs (ID: 1263172-11)

Further information: User's Manual for Cycle Programming (ID: 1096886-16)

Machining cycles

- Cycle **224 DATAMATRIX CODE PATTERN** (ISO: **G224**)
This cycle allows the control to convert a text into a machine-readable Data Matrix code. The Data Matrix code functions as a point pattern for a previously defined machining cycle.
- Cycle **271 OCM CONTOUR DATA** (ISO: **G271**, option 167)
This cycle allows you to define the machining information for the OCM cycles. You can define the first pocket of the contour description as an open frame. As a result, the downfeed is performed outside of the material during machining.
- Cycle **272 OCM ROUGHING** (ISO: **G272**, option 167)
This cycle allows the control to machine the defined contour using trochoidal milling. The control precisely adheres to the programmed path overlap, while keeping a constant tool contact angle and depth of cut.
- Cycle **273 OCM FINISHING FLOOR** (ISO: **G273**, option 167)
This cycle allows the control to finish the contour floor with a constant path overlap. The finishing allowance for the side is left over.
- Cycle **274 OCM FINISHING SIDE** (ISO: **G274**, option 167)
This cycle enables finishing along the contour. The control approaches the contour tangentially and machines every subcontour separately. The control does not take into account the finishing allowance defined for the floor in Cycle **271 (G271)**, but moves to full depth.

Touch probe cycles (option 17)

Cycle **238 MEASURE MACHINE STATUS** (ISO: **G238**, option 155)

This cycle allows the control to measure various components defined to be monitored. The machine manufacturer defines the scope and the sequence of the cycle. You cannot change the feed rate during measurement.

If you define a value of 0 in the parameter **Q570 MODE**, then the control performs a dry run. During the dry run the control does not measure any values and you can change the feed rate.

Program functions

- **FUNCTION CORRDATA** allows you to activate a line of the compensation table. The compensation remains effective until the next tool change or the end of the program.
- You can also use subprograms and QS parameters within the **CONTOUR DEF** function.
- **FUNCTION TCPM** (option 9) is available in ISO programming mode. **FUNCTION TCPM** is an improvement over **M128** and allows you to define the behavior of the control during the positioning of the rotary axes.

Miscellaneous functions

During retraction with **M140 MB MAX**, the control takes into account the safety clearances that can be defined by the machine manufacturer for software limit switches and collision objects. The control reduces the retraction movements by the clearances and stops before the software limit switches.

Programming Q parameters

- The **FN 18: SYSREAD (ISO: D18)** functions have been enhanced:
 - **FN 18: SYSREAD (D18) ID15**
 - **NR10**: contents of a Q parameter
 - **NR11**: contents of a QL parameter
 - **NR12**: contents of a QR parameter
 - **FN 18: SYSREAD (D18) ID35 NR2**: active radius compensation
 - **FN 18: SYSREAD (D18) ID210 NR10**: type of active tilting function
 - **FN 18: SYSREAD (D18) ID10010 NR1** and **NR2**: information about the current main program or the called NC program as a text variable
 - **IDX0**: entire file path
 - **IDX1**: directory path
 - **IDX2**: file name
 - **IDX3**: file type
 - **FN 18: SYSREAD (D18) ID10015**
 - **NR20**: contents of a QS parameter
 - **NR30**: contents of a QS parameter (all characters, except the letters and numbers, are replaced with an underscore (_) character)
- If you use the **SQL EXECUTE** function and the **CREATE TABLE** statement to create a table, then you define the sequence of the columns with the **AS SELECT** statement.

Calling an external NC program

The **SELECT COMPENS. TABLE** soft key has been added to the soft-key row of the **PGM CALL** functions. This soft key activates the **SEL CORR-TABLE** function that allows you to activate a compensation table for the NC program.

Test run (option 20)

- In **Test Run** operating mode, the control uses the active preset.
- The **RESET THE PRESET** soft key has been added to the **BLANK IN WORK SPACE** menu. This soft key allows you to set the values of the active preset in the principal axis to 0 for the simulation.

Program run

- The control saves active NC programs only up to a maximum size of 10 MB each to a service file. NC programs larger than that are not saved.
- The **SELECT COMPENS. TABLES** soft key has been added to the operating modes **Program run, single block** and **Program run, full sequence**. This soft key allows you to open and edit the active datum table and the active compensation tables.
- In the operating modes **Program run, single block** and **Program run, full sequence**, the **ACTUAL POSITION CAPTURE** key allows you to load the current position values of an axis into the datum table.

Status displays

- The control displays a corresponding icon in the general status display when tool radius compensation is active.
- The control shows the **POS HR** tab of the additional status display without the option 44 being enabled. The control shows the active maximum values of handwheel superimpositioning from **M118** (option 21).

Moving the machine axes

The HR 520 and HR 550 FS handwheels display the offset of handwheel superimpositioning from **M118** (option 21). The handwheels also show the active setting of the 3-D ROT menu (option 8).

Tilting the working plane (option 8)

The **3-D ROT: AXIS ANGLE SPA. ANGLE** soft key has been added to the 3-D ROT menu. This soft key allows you to select whether the control displays the defined axis values or the current spatial angles.

Tables

The preset table contains default values only in the columns **SPA**, **SPB**, and **SPC**. The machine manufacturer can define default values for the other columns.

Tools

- Compensation tables allow you to compensate for tools during program run without having to edit the NC program or the tool tables. The compensation table *.tco is effective in the tool coordinate system as an alternative to the compensation in the tool call.
- The **ACTUAL POSITION CAPTURE** soft key allows you to load the actual position of the tool axis into the form view of the tool management as the tool length.

Error messages and help system

- In the error window, the **ACTIVATE AUTOMATIC SAVING** soft key has been added to **MORE FUNCTIONS**. This soft key allows you to define up to five error numbers. The control will automatically create a service file upon occurrence of these error numbers.
- In the optional machine parameter **CfgClearError**, the machine manufacturer defines whether the control automatically clears warning and error messages when an NC program is selected or restarted.

HEROS functions

- When user administration is active, the file manager displays the **public** directory that can be accessed by every user.
When you place the cursor on the **public** directory, the control shows the **ADVANCED ACCESS RIGHTS** soft key. This soft key allows the owner of a file to define the access rights of the following users:
 - Owner
 - Group
 - Other users
- The **useradmin**, **oem**, and **sys** users can deactivate the user administration.

Machine-specific user parameters

- The optional machine parameter **applyCfgLanguage** (no. 101305) allows you to define whether the HEROS operating system adopts the conversational language defined in machine parameter **ncLanguage** (no. 101301) during booting. If you activate this function, you can change the conversational language only in the machine parameters.
- The optional machine parameter **extendedDiagnosis** (no. 124204) allows you to define whether the control saves graphics journal data after a restart. This data is used to diagnose graphics problems.

8.3 Modified functions

Further information: User's Manual for Conversational Programming (ID: 1096883-17)

Further information: User's Manual for Setup, Testing and Running NC Programs (ID: 1263172-11)

Further information: User's Manual for Cycle Programming (ID: 1096886-16)

Machining cycles

- In Cycle **225 ENGRAVING** (ISO: **G225**), you can program system variables that contain the following information for the current main program and the NC program called:
 - Entire file path
 - Directory path
 - File name
 - File type
- If, in Cycle **233 FACE MILLING** (ISO: **G233**, option 19), you program a limit that is perpendicular to the milling direction **Q350**, then the control adds the tool radius to the length of the surface in the unlimited direction. As a result, the control completely machines the defined surface without leaving behind any residual material, as would be caused by the tool radius. If the parameter **Q220** (corner radius) is defined, then the control adds both the tool radius and this value to the length of the surface.
- Cycles **205 UNIVERSAL PECKING** (ISO: **G205**) and **241 SINGLE-LIP D.H.DRLNG** (ISO: **G241**, option 19) check the parameter **Q379 STARTING POINT**. If the value of the starting point is equal to or greater than the value of the parameter **Q201 DEPTH**, then the control issues an error message.
- The machine manufacturer can hide the Cycle **239 ASCERTAIN THE LOAD** (ISO: **G239**, option 143).

Touch probe cycles (option 17)

If you run the Cycles **1410 PROBING ON EDGE** (ISO: **G1410**), **1411 PROBING TWO CIRCLES** (ISO: **G1411**), and **1420 PROBING IN PLANE** (ISO: **G1420**) in semiautomatic mode, then you can use a handwheel for prepositioning and position to a clearance height at the end of the cycle.

Program functions

You can limit the feed rate of the compensating movements in **FUNCTION TCPM** (option 9).

Miscellaneous functions

- If you program a precision stop of rotary axes using **M134** or **M135**, the control no longer displays an error message. The control ignores these miscellaneous functions.
- The number range for the machine manufacturer's miscellaneous functions has been increased from 1999 to 9999.

Programming Q parameters

- The **FN 10** function also allows you to check QS parameters and texts for inequalities.
- You can use combined QS parameters in the **SQL EXECUTE** and **SQL SELECT** functions.

Test run (option 20)

- If you set the **MEASURING** soft key to **ON**, then the control displays the following additional information:
 - Surface orientation of the current position
 - Workpiece number
 - Workpiece name
 - Note during machining at rapid traverse, thread cutting cycle, or blank form update
- The **BLANK IN WORK SPACE** menu provides a soft key that allows you to load the current machine status. The control loads the following information in addition to the active preset:
 - Active kinematics
 - Active traverse ranges
 - Active machining mode
 - Active traverse limits

File management

Any display filter you have set in the file manager will be retained even after a control restart.

Program run

- The machine manufacturer defines the sequence for traversing the axes when returning to the contour.
- The control considers the functions **FN 27: TABWRITE** (ISO: D27) and **FUNCTION FILE** only in the operating modes **Program run, single block** and **Program run, full sequence**.

Status displays

- On a 19" screen, the control displays up to ten axes on the **Overview** tab of the additional status display.
- The **CM** and **CM Detail** tabs of the additional status display have been renamed as **MON** and **MON Detail**, respectively (option 155).
- The control also displays the number of repetitions on the **Overview** and **LBL** tabs of the additional status display after an internal stop.

Moving the machine axes

The smallest definable speed level on handwheels with display has been changed from 0.1 % to 0.01 % of the maximum handwheel speed.

Manual touch probe cycles (option 17)

If, during probing, the axis angles do not match the tilt angles, then the **Working plane is inconsistent** menu allows you to tilt the rotary axes to the angle defined in the 3-D ROT menu (option 8). For this purpose, the **ALIGN ROT. AXES** soft key has been added.

Tables

The input range of the columns **SPA**, **SPB**, **SPC**, **A_OFFS**, **B_OFFS**, and **C_OFFS** in the preset table has been changed from +/- 999.9999999° to +/-99 999.99999°.

Tools

The form view of the tool management shows only those input fields that are needed for the selected tool type.

MOD functions

- If the control is accessed from external, it displays a corresponding icon in the header.
- Limits that have been defined in the **Traverse limits** MOD function are also effective for modulo axes.
- **Program run** in the **Machine times** MOD area shows only the times at which at least one axis was moving during program run.

HEROS functions

- When user administration is active, the **Liberating motion** mode requires the NC.OPModeManual permission (i.e., at least the role of **NC.Programmer**).
- The machine manufacturer can define a path for saving the values of the QR parameters. If the values are on the **TNC** drive, you can use the **NC/PLC Backup** HEROS function to back up the QR parameters.

9

Software 81760x-08

9.1 New software options

Further information: User's Manual for Setup, Testing and Running NC Programs - 1263172-12

HEIDENHAIN OPC UA NC Server (options 56 to 61)

OPC UA provides a standardized interface for secure data exchange between products independent of the manufacturer. HEIDENHAIN provides the **HEIDENHAIN OPC UA NC Server** for data exchange with the control. These software options allow you to create up to six parallel client connections.

The **Connection Assistant** function has been added to the HEROS menu for setting up the connection. If user administration is active, you link the connections to specific users.

As part of the **HEIDENHAIN OPC UA NC Server** (options 56 to 61), the machine parameter **CfgMachineInfo** (no. 131700), which allows you to define information about the machine, has been added.

Renamed software options

Option 146 has been renamed as **Machine Vibration Control MVC**

The Frequency Shaping Control (**FSC**) function has been added; it allows the control to suppress low-frequency machine vibrations.

9.2 New functions

Further information: User's Manual for **Klartext Programming** - 1096883-18

Further information: User's Manual for **Setup, Testing and Running NC Programs** - 1263172-12

Further information: User's Manual for **Programming of Machining Cycles** - 1303427-10

Further information: User's Manual for **Programming of Measuring Cycles for Workpieces and Tools** - 1303435-10

NC fundamentals

The **BLK FORM FILE** function allows you to use STL files to define the workpiece blank, and optionally the finished part, by specifying the path of the files. This enables you to use, e.g., 3-D models from the CAD system in your NC program

Machining cycles

- Cycle **277 OCM CHAMFERING** (ISO: **G277**, option 167)
This cycle allows the control to deburr contours that were last defined, roughed, or finish-machined with other OCM cycles.
- Cycle **1271 OCM RECTANGLE** (ISO: **G1271**, option 167)
This cycle allows you to define a rectangle that is then used as a pocket, island, or boundary for face milling in conjunction with other OCM cycles.
- Cycle **1272 OCM CIRCLE** (ISO: **G1272**, option 167)
This cycle allows you to define a circle that is then used as a pocket, island, or boundary for face milling in conjunction with other OCM cycles.
- Cycle **1273 OCM SLOT / RIDGE** (ISO: **G1273**, option 167)
This cycle allows you to define a slot that is then used as a pocket, island, or boundary for face milling in conjunction with other OCM cycles.
- Cycle **1278 OCM POLYGON** (ISO: **G1278**, option 167)
This cycle allows you to define a polygon that is then used as a pocket, island, or boundary for face milling in conjunction with other OCM cycles.
- Cycle **1281 OCM RECTANGLE BOUNDARY** (ISO: **G1281**, option 167)
This cycle allows you to define a rectangular boundary for islands or open pockets that you previously programmed with the standard OCM forms.
- Cycle **1282 OCM CIRCLE BOUNDARY** (ISO: **G1282**, option 167)
This cycle allows you to define a circular boundary for islands or open pockets that you previously programmed with the standard OCM forms.

Program functions

- **FUNCTION MODE SET** allows you to activate settings defined by the machine tool builder (e.g., changes to the range of traverse) from within the NC program
- The function **PRESET SELECT** allows you to activate a preset from the preset table. You can choose to retain active transformations and select the preset to which the function should apply.
- The function **PRESET COPY** allows you to copy a preset defined in the preset table to another row. You can optionally activate the copied preset and retain the active transformations.
- The function **PRESET CORR** allows you to correct the active preset.
- The function **OPEN FILE** allows the control to open different file formats (e.g., PNG files) using a suitable additional tool.
- The function **POLARKIN** allows you to activate a polar kinematic model. In a polar kinematic model, the control performs movements using a rotary axis and two linear axes. You define the positioning behavior of the rotary axis, as well as whether machining is allowed at the center of rotation of the rotary axis.
- The function **TABDATA** allows you to access the tool table and the compensation tables (*.tco and *.wco) during program run. In order to access the compensation tables, you need to activate them.
 - The function **TABDATA READ** allows you to read a value from a table and save it to a Q, QL, QR, or QS parameter.
 - The function **TABDATA WRITE** allows you to write a value from a Q, QL, QR, or QS parameter into a table.
 - The function **TABDATA ADD** allows you to add a value from a Q, QL, or QR parameter to a value contained in a table.
- The function **MONITORING** allows you to visualize the monitoring of a defined machine component.
- The **APPLY FILE NAME** soft key has been added to the selection window provided by the **SELECT FILE** soft key. If the called file is located in the same directory as the file you are calling it from, pressing this soft key loads the name of the file without its path.

Programming Q parameters

- In the mask file of the **FN 16: F-PRINT** (ISO: **D16**) function, you can define whether the control shows or hides blank lines for undefined QS parameters.
- The **FN 18: SYSREAD** functions (ISO:D18) have been enhanced:
 - **FN 18: SYSREAD (D18) ID50**: values in the tool table
 - **NR45**: value in the **RCUTS** column
 - **NR46**: value in the **LU** column
 - **FN 18: SYSREAD (D18) ID950**: values in the tool table of the current tool
 - **NR45**: value in the **RCUTS** column
 - **NR46**: value in the **LU** column
 - **NR47**: value in the **RN** column
 - **FN 18: SYSREAD (D18) ID1070 NR1**: active feed-rate limit through the **F MAX** soft key
- The function **SYSSTR(ID10321 NR20)** allows you to determine the number of the current week in accordance with ISO 8601

Programming aids

The control offers an **OCM cutting data calculator** with which you can determine the optimum cutting data for the **272 OCM ROUGHING** cycle (ISO: **G272**, option 167). Press the **OCM CUTTING DATA** soft key to open the cutting data calculator during cycle definition. You can load the results directly into the cycle parameters.

Test run (option 20)

- If you use **TARGET** in the **BLK FORM FILE** function to define a finished part, you can show or hide it by soft key in the **Test Run** operating mode (option 20)
- In **Test Run** operating mode, you can use the **EXPORT WORKPIECE** soft key to export the current status of the motion simulation as a 3-D model in STL format.
- In **Test Run** operating mode, the control provides an extended collision monitoring function that checks for collisions between the workpiece and the tool or tool holder. You can activate extended collision monitoring by soft key.

File management

- The control supports USB data media with the NTFS file system.
- The control includes the additional tool **Parole**, which allows you to open video files.

CAD Viewer

- When you double-click a layer in the **CAD-Viewer**, the control marks the first contour element of this layer.
- You can transfer the data from the CAD Import clipboard to an NC program as well as to other applications (e.g., **Leafpad**).

Status displays

- If a feed-rate limit has been activated with the **F MAX** soft key, the control displays an exclamation mark after the feed-rate value in the general status display.
- When the **PARAXCOMP DISPLAY** function is active, the control displays a corresponding icon in the general status display.
- When the **PARAXCOMP MOVE** function is active, the control displays a corresponding icon in the general status display.
- When the functions **PARAXMODE** oder **POLARKIN** are active, the control displays a corresponding icon in the general status display.

Tools

- In the **RCUTS** column of the tool table, you define the front-face cutting width of a tool (e.g., for indexable inserts).
- You define the usable length of a tool in the **LU** column of the tool table. The usable length limits the plunging depth of the tool in cycles.
- You define the neck radius of the tool in the **RN** column of the tool table. This allows the control to display the tool neck correctly in the simulation (e.g., for side milling cutters).
- You can use M3D and STL files (e.g., from the CAD system) as tool carrier files.

Error messages and help system

If, after a hardware change or an update, an error occurs when the control is booting, the control will automatically open the error window and display a question-type error. The control displays soft keys providing different response options.

MOD functions

- A link to the **Firewall settings** HEROS function has been added to the **External access** MOD function.
- A link to the **OPC UA NC Server - License Settings** HEROS function (options 56 to 61) has been added within the **External access** MOD function.
- If the machine tool builder has defined the parameter **CfgOemInfo** (no. 131700), then the control displays the **Info about machine manufacturer** area in the **General information** MOD group.
- If the machine operator has defined the **CfgMachineInfo** parameter (no. 131600), the control shows the **Machine information** area within the **General information** MOD group.

Remote Desktop Manager (option 133)

Remote Desktop Manager (option 133) allows you to create private connections if user administration is active. Private connections are visible to and usable by the creator only.

HEROS functions

- When user administration is active, the control automatically disables the LSV2 connections of the serial interfaces (COM1 and COM2) for security purposes.
- When user administration is active, you can create user-specific private network drive connections. **Single Sign On** allows you to connect to an encrypted network drive while logging on to the control.
- When configuring the user administration, you can use the **Autologin** function to define a user who will automatically be logged on by the control during booting.

Machine-specific user parameters

The machine parameter **CfgTTRectStylus** (no. 114300) has been added. This parameter allows you to define settings for a tool touch probe with a cuboid probe contact.

9.3 Modified functions

Further information: User's Manual for **Klartext Programming** - 1096883-18

Further information: User's Manual for **Setup, Testing and Running NC Programs** - 1263172-12

Further information: User's Manual for **Programming of Machining Cycles** - 1303427-10

Further information: User's Manual for **Programming of Measuring Cycles for Workpieces and Tools** - 1303435-10

NC fundamentals

You can use the **RND** element (ISO: **G24**) to connect circles that are perpendicular to the machining plane instead of lying in the machining plane.

Machining cycles

- Cycle **225 ENGRAVING** (ISO: **G225**) allows you to use system variables to engrave the number of the current week.
- Cycles **202 BORING** (ISO: **G202**) and **204 BACK BORING** (ISO: **G204**, option 19) restore the spindle status after machining to that which was active before the cycle.
- If the defined usable length in column **LU** of the tool table is less than the depth, the control displays an error message.
The following cycles monitor the usable length **LU**:
 - All cycles for drilling and boring
 - All cycles for tapping
 - All cycles for the machining of pockets and studs
 - Cycle 22 **ROUGH-OUT** (ISO: **G122**, option 19)
 - Cycle 23 **FLOOR FINISHING** (ISO: **G123**, option 19)
 - Cycle 24 **SIDE FINISHING** (ISO: **G124**, option 19)
 - Cycle 233 **FACE MILLING** (ISO: **G233**, option 19)
 - Cycle 272 **OCM ROUGHING** (ISO: **G272**, option 167)
 - Cycle 273 **OCM FINISHING FLOOR** (ISO: **G273**, option 167)
 - Cycle 274 **OCM FINISHING SIDE** (ISO: **G274**, option 167)
- Cycles **251 RECTANGULAR POCKET** (ISO: **G251**), **252 CIRCULAR POCKET** (ISO: **G252**, option 19), and **272 OCM ROUGHING** (ISO: **G272**, option 167) take into consideration a cutting width defined in the column **RCUTS** when calculating the plunging path.
- Cycles **208 BORE MILLING** (ISO: **G208**), **253 SLOT MILLING** (ISO: **G208**), and **254 CIRCULAR SLOT** (ISO: **G254**, option 19) monitor a cutting width defined in the column **RCUTS** of the tool table. If the center of a tool that is not a center-cut tool would contact the workpiece surface, the control issues an error message.
- The machine manufacturer can hide Cycle **238 MEASURE MACHINE STATUS** (ISO: **G238**, option 155).
- An input value of 2 has been added to parameter **Q569 OPEN BOUNDARY** in Cycle **271 OCM CONTOUR DATA** (ISO: **G271**, option 167). The control uses this value to interpret the first contour within the **CONTOUR DEF** function as a boundary block of a pocket.
- Cycle **272 OCM ROUGHING** (ISO: **G272**, option 167) was expanded:
 - The parameter **Q576 SPINDLE SPEED** allows you to define a spindle speed for the roughing tool.
 - The parameter **Q579 PLUNGING FACTOR S** allows you to define a factor for the spindle speed during plunging.
 - Use the parameter **Q575 INFEEED STRATEGY** to define whether the control machines the contour from top to bottom or vice versa.
 - The maximum input range of parameter **Q370 TOOL PATH OVERLAP** has been changed. Old: from 0.01 to 1. New: from 0.04 to 1.99.
 - If helical plunging is not possible, the control attempts a reciprocating plunging motion with the tool.

- Cycle **273 OCM FINISHING FLOOR** (ISO: **G273**, option 167) was expanded.

The following parameters have been added:

- **Q595 STRATEGY**: machining with equal path distances or constant tool angle
- **Q577 APPROACH RADIUS FACTOR**: factor for the tool radius in order to adapt the approach radius

Touch probe cycles (option 17)

- Cycles **480 CALIBRATE TT** (ISO: **G480**) and **484 CALIBRATE IR TT** (ISO: **G484**, option 17) can be used to calibrate a tool touch probe with cuboid probe contacts.
- For rotating tools, Cycle **483 MEASURE TOOL** (ISO: **G483**, option 17) first measures the tool length and then the tool radius.
- By default, Cycles **1410 PROBING ON EDGE** (ISO: **G1410**) and **1411 PROBING TWO CIRCLES** (ISO: **G1411**, option 17) calculate the basic rotation in the input coordinate system (I-CS). If the axis angles and tilt angles don't coincide, the cycles calculate the basic rotation in the workpiece coordinate system (W-CS).

Miscellaneous functions

- The **M109** function allows the control to keep the feed rate constant at the cutting edge, including during approach and departure.
- The function **M120** (option 21) for look-ahead calculation of a radius-compensated contour is no longer reset by milling cycles (option 19)

Programming Q parameters

- You can use UTF-8 character encoding in the mask file of **FN 16: F-PRINT** (ISO: **D16**)
- The priority of arithmetic operations has been changed in the Q parameter formula

Programming aids

- Scrolling in the structure window works in the same way as scrolling in the NC program You can define the position of the active structure block by soft key.
- The control uses the active unit of measure (mm or inches) for calculations in the cutting data calculator.

Test run (option 20)

The control uses hatch marking to depict threads in the simulation.

CAD Viewer

Path finding between individual drilling positions has been optimized in the **CAD-Viewer**

Batch Process Manager (option 154)

In the operating modes **Program run, single block** and **Program run, full sequence**, the **Batch Process Manager** (option 154) shows up to two statuses next to each other in the first column.

Program run

- In **Program run, single block** operating mode, the control interprets the workpiece blank definition as a single NC block only.
- The control displays the tool index in the block scan pop-up window if needed.
- The control takes into account manual axes when returning to the contour.

Pallets (option 22)

In pallet tables (option 22), the control can also open NC programs containing blank spaces.

Status displays

- When the **PARAXCOMP DISPLAY** function or the **PARAXCOMP MOVE** function is active, the control displays **(D)** or **(M)** after the respective axis designations on the **Overview** tab and the **POS** tab of the additional status display.
- On the **FS** tab of the additional status display, the control displays the active limits defined by the specific safety-related operating modes for every axis.
- On the **TT** tab of the additional status display, the control displays the tilt angle of the tool touch probe as well as information about the cuboid probe contacts.
- In **Test Run** operating mode, the control displays the **M** tab of the additional status display when the screen layout **PROGRAM + STATUS** is active.

Moving the machine axes

- When you activate a handwheel with display, the control automatically activates the override potentiometer of the handwheel.
- In the operating modes **Manual Operation** and **Positioning w/ Manual Data Input**, you can activate a handwheel with display while a macro or a manual tool change is being executed.
- You can activate or deactivate the **F MAX** soft key for reducing the feed rate. The defined value is retained.

Manual touch probe cycles (option 17)

By default, the control calculates the basic rotation in the input coordinate system (I-CS). If the axis angles and tilt angles don't coincide, the control calculates the basic rotation in the workpiece coordinate system (W-CS).

Tables

In the compensation tables (*.tco and *.wco), the input range for all columns containing numerical values has been changed from +/- 999.999 to +/- 999.9999

Error messages and help system

The **FILTER** soft key in the error window allows the control to classify both error messages and warnings. As a result, the list of pending messages becomes shorter and easier to read.

MOD functions

From within the **Diagnostic functions** MOD group, you can access **TNCdiag** and **Hardware configuration** without a code number.

Remote Desktop Manager (option 133)

The name of a connection in the **Remote Desktop Manager** (option 133) must not contain characters other than letters, numbers, and underscores.

HEROS functions

- If you enter a password or code number with Caps Lock active, then the control issues a message.
- The **HEIDENHAIN OPC UA NC Server** enables you to access the directories **TNC:** and **PLC:**, including while the NC software is switched off. The contents displayed vary depending on the assigned user's permissions.
- If, when configuring the user administration, you use the **Connection to Windows domain** function, the **Use LDAPs** check box allows you to set up a secure connection.
- If a remote log-in takes place while user administration is inactive, for example via SSH, the control automatically assigns the **HEROS.LegacyUserNoCtrlfct** role.
- When user administration is active, the NC.SetupProgramRun permission is required for the **ACC** (option 145) functions.
- If you deactivate the user administration and select the **Delete existing user databases** check box, the control also deletes the .home folder in the **TNC:** directory.

Machine-specific user parameters

The machine parameter **spindleDisplay** (no. 100807) has been enhanced. The control can also display the spindle position on the **Overview** tab of the additional status display when the spindle is in jog mode.

HEIDENHAIN

DR. JOHANNES HEIDENHAIN GmbH

Dr.-Johannes-Heidenhain-Straße 5

83301 Traunreut, Germany

☎ +49 8669 31-0

FAX +49 8669 32-5061

E-mail: info@heidenhain.de

Technical support FAX +49 8669 32-1000

Measuring systems ☎ +49 8669 31-3104

E-mail: service.ms-support@heidenhain.de

NC support ☎ +49 8669 31-3101

E-mail: service.nc-support@heidenhain.de

NC programming ☎ +49 8669 31-3103

E-mail: service.nc-pgm@heidenhain.de

PLC programming ☎ +49 8669 31-3102

E-mail: service.plc@heidenhain.de

APP programming ☎ +49 8669 31-3106

E-mail: service.app@heidenhain.de

www.heidenhain.de

