



TNC 640

Overview of New and Modified Software Functions

NC Software
34059x-02 to 34059x-17

About this document

This document describes the new and modified software functions of the TNC 640. 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 34059x-02	19
2	Software 34059x-04	27
3	Software 34059x-05	35
4	Software 34059x-06	45
5	Software 34059x-07	53
6	Software 34059x-08	65
7	Software 34059x-09	79
8	Software 34059x-10	89
9	Software 34059x-11	101
10	Software 34059x-16	111
11	Software 34059x-17	121

1	Software 34059x-02.....	19
1.1	New software options.....	20
1.1.1	Dynamic Collision Monitoring (DCM) (option 40).....	20
1.1.2	CAD Import (option 42).....	20
1.1.3	Adaptive Feed Control (AFC) (option 45).....	20
1.1.4	Remote Desktop Manager (option 133).....	20
1.1.5	Cross Talk Compensation (CTC) (option 141).....	20
1.1.6	Position Adaptive Control (PAC) (option 142).....	20
1.1.7	Load Adaptive Control (LAC) (option 143).....	20
1.1.8	Motion Adaptive Control (MAC) (option 144).....	21
1.1.9	Active Chatter Control (ACC) (option 145).....	21
1.2	New functions.....	21
1.2.1	Machining cycles.....	21
1.2.2	Touch probe cycles.....	22
1.2.3	Miscellaneous functions.....	22
1.2.4	Program functions.....	22
1.2.5	Programming Q parameters.....	23
1.2.6	File management.....	23
1.2.7	Moving the machine axes.....	23
1.2.8	Manual touch probe cycles.....	23
1.2.9	Accessories.....	23
1.2.10	MOD functions.....	23
1.2.11	Machine-specific user parameters.....	24
1.3	Modified functions.....	24
1.3.1	NC fundamentals.....	24
1.3.2	Machining cycles.....	24
1.3.3	Touch probe cycles.....	24
1.3.4	Program functions.....	25
1.3.5	Programming Q parameters.....	25
1.3.6	Programming aids.....	25
1.3.7	Program run.....	25
1.3.8	Positioning w/ Manual Data Input operating mode.....	25
1.3.9	Status displays.....	25
1.3.10	Moving the machine axes.....	25
1.3.11	Manual touch probe cycles.....	26
1.3.12	Tilting the working plane (option 8).....	26
1.3.13	Tools.....	26
1.3.14	Machine-specific user parameters.....	26

2	Software 34059x-04.....	27
2.1	New software options.....	28
2.1.1	Synchronizing functions (option 135).....	28
2.1.2	Active Vibration Damping (AVD) (option 146).....	28
2.1.3	Conversational languages.....	28
2.2	New functions.....	28
2.2.1	NC fundamentals.....	28
2.2.2	Machining cycles.....	29
2.2.3	Touch probe cycles.....	29
2.2.4	Program functions.....	29
2.2.5	Adaptive Feed Control (AFC) (option 45).....	29
2.2.6	Programming Q parameters.....	30
2.2.7	Programming aids.....	30
2.2.8	Test run.....	30
2.2.9	Program run.....	30
2.2.10	Active Chatter Control (ACC) (option 145).....	31
2.2.11	Tables.....	31
2.2.12	Tools.....	31
2.2.13	MOD functions.....	31
2.3	Modified functions.....	31
2.3.1	NC fundamentals.....	31
2.3.2	Machining cycles.....	32
2.3.3	Touch probe cycles.....	33
2.3.4	Miscellaneous functions.....	33
2.3.5	Programming Q parameters.....	33
2.3.6	Programming aids.....	33
2.3.7	DXF Converter (option 42).....	33
2.3.8	Dynamic Collision Monitoring (DCM) (option 40).....	33
2.3.9	Moving the machine axes.....	33
2.3.10	Manual touch probe cycles.....	33
2.3.11	Tables.....	33
2.3.12	Tools.....	33
2.3.13	Error messages and help system.....	34
2.3.14	MOD functions.....	34
2.3.15	Machine-specific user parameters.....	34

3	Software 34059x-05.....	35
3.1	New software options.....	36
3.1.1	KinematicsComp (option 52).....	36
3.1.2	Advanced Spindle Interpolation (option 96).....	36
3.1.3	Spindle Synchronism (option 131).....	36
3.1.4	Renamed software options.....	36
3.2	New functions.....	36
3.2.1	NC fundamentals.....	36
3.2.2	Machining cycles.....	37
3.2.3	Program functions.....	37
3.2.4	Dynamic Collision Monitoring (DCM) (option 40).....	37
3.2.5	Calling an external NC program.....	37
3.2.6	Programming Q parameters.....	38
3.2.7	File management.....	38
3.2.8	CAD Viewer.....	38
3.2.9	Manual touch probe cycles.....	38
3.2.10	Tables.....	38
3.2.11	Tools.....	39
3.2.12	MOD functions.....	39
3.2.13	HEROS functions.....	39
3.2.14	Machine-specific user parameters.....	40
3.3	Modified functions.....	40
3.3.1	NC fundamentals.....	40
3.3.2	Machining cycles.....	41
3.3.3	Touch probe cycles.....	41
3.3.4	Miscellaneous functions.....	42
3.3.5	Programming Q parameters.....	42
3.3.6	Programming aids.....	42
3.3.7	File management.....	42
3.3.8	DXF Converter (option 42).....	42
3.3.9	Program run.....	42
3.3.10	Status displays.....	42
3.3.11	Tilting the working plane (option 8).....	42
3.3.12	Manual touch probe cycles.....	43
3.3.13	Tables.....	43
3.3.14	Tools.....	43
3.3.15	Machine-specific user parameters.....	43

4	Software 34059x-06	45
4.1	New software options	46
4.1.1	Camera-based Visual Setup Control (VSC) (option 136)	46
4.2	New functions	46
4.2.1	Machining cycles	46
4.2.2	Touch probe cycles	46
4.2.3	Program functions	46
4.2.4	Adaptive Feed Control (AFC) (option 45)	46
4.2.5	Programming Q parameters	47
4.2.6	Programming aids	47
4.2.7	File management	47
4.2.8	DXF Converter (option 42)	47
4.2.9	Program run	47
4.2.10	Pallets	47
4.2.11	Moving the machine axes	47
4.2.12	Manual touch probe cycles	48
4.2.13	Tables	48
4.2.14	Tools	48
4.2.15	Machine-specific user parameters	48
4.3	Modified functions	48
4.3.1	NC fundamentals	48
4.3.2	Machining cycles	49
4.3.3	Touch probe cycles	49
4.3.4	Miscellaneous functions	50
4.3.5	DXF Converter (option 42)	50
4.3.6	Program run	50
4.3.7	Pallets	50
4.3.8	Positioning w/ Manual Data Input operating mode	50
4.3.9	Status displays	50
4.3.10	Manual touch probe cycles	50
4.3.11	Tables	51
4.3.12	Tools	51
4.3.13	MOD functions	51

5 Software 34059x-07.....	53
5.1 New software options.....	54
5.1.1 3-D radius compensation depending on the tool’s contact angle (option 92).....	54
5.2 New functions.....	54
5.2.1 NC fundamentals.....	54
5.2.2 Touch probe cycles.....	54
5.2.3 Program functions.....	54
5.2.4 Adaptive Feed Control (AFC) (option 45).....	55
5.2.5 Programming Q parameters.....	55
5.2.6 Programming aids.....	55
5.2.7 Test run.....	56
5.2.8 File management.....	56
5.2.9 Program run.....	56
5.2.10 Positioning w/ Manual Data Input operating mode.....	56
5.2.11 Status displays.....	57
5.2.12 Moving the machine axes.....	57
5.2.13 Tilting the working plane (option 8).....	57
5.2.14 Tools.....	57
5.2.15 Error messages and help system.....	57
5.2.16 HEROS functions.....	57
5.2.17 Machine-specific user parameters.....	58

5.3	Modified functions.....	58
5.3.1	NC fundamentals.....	58
5.3.2	Machining cycles.....	59
5.3.3	Touch probe cycles.....	60
5.3.4	Program functions.....	60
5.3.5	Programming Q parameters.....	60
5.3.6	Calling an external NC program.....	60
5.3.7	Programming aids.....	60
5.3.8	Test run.....	61
5.3.9	File management.....	61
5.3.10	DXF Converter (option 42).....	61
5.3.11	Dynamic Collision Monitoring (DCM) (option 40).....	61
5.3.12	Program run.....	61
5.3.13	Pallets.....	61
5.3.14	Status displays.....	61
5.3.15	Camera-based monitoring of setup situation (VSC) (option 136).....	62
5.3.16	Moving the machine axes.....	62
5.3.17	Manual touch probe cycles.....	62
5.3.18	Tilting the working plane (option 8).....	62
5.3.19	Tables.....	62
5.3.20	Tools.....	62
5.3.21	MOD functions.....	62
5.3.22	HEROS functions.....	63

6	Software 34059x-08.....	65
6.1	New software options.....	66
6.1.1	Global Program Settings (option 44).....	66
6.1.2	Batch Process Manager (option 154).....	66
6.1.3	Renamed software options.....	66
6.2	New functions.....	66
6.2.1	NC fundamentals.....	66
6.2.2	Machining cycles.....	67
6.2.3	Touch probe cycles.....	67
6.2.4	Program functions.....	67
6.2.5	Adaptive Feed Control (AFC) (option 45).....	68
6.2.6	Miscellaneous functions.....	68
6.2.7	Programming Q parameters.....	69
6.2.8	Calling an external NC program.....	69
6.2.9	Programming aids.....	69
6.2.10	Test run.....	70
6.2.11	File management.....	70
6.2.12	CAD Viewer.....	70
6.2.13	Pallets.....	70
6.2.14	Status displays.....	70
6.2.15	Manual touch probe cycles.....	71
6.2.16	Tables.....	71
6.2.17	Tools.....	71
6.2.18	Accessories.....	71
6.2.19	MOD functions.....	71
6.2.20	HEROS functions.....	72
6.2.21	Machine-specific user parameters.....	72

6.3	Modified functions.....	72
6.3.1	NC fundamentals.....	72
6.3.2	Machining cycles.....	73
6.3.3	Touch probe cycles.....	74
6.3.4	Program functions.....	74
6.3.5	Miscellaneous functions.....	74
6.3.6	Programming Q parameters.....	75
6.3.7	Programming aids.....	75
6.3.8	Test run.....	75
6.3.9	File management.....	75
6.3.10	Program run.....	75
6.3.11	Status displays.....	76
6.3.12	Camera-based monitoring of setup situation (VSC) (option 136).....	76
6.3.13	Moving the machine axes.....	76
6.3.14	Tables.....	76
6.3.15	Tools.....	76
6.3.16	Error messages and help system.....	76
6.3.17	Remote Desktop Manager (option 133).....	77

7	Software 34059x-09	79
7.1	New software options	80
7.1.1	State Reporting Interface (option 137)	80
7.1.2	Monitoring of configured machine components for overload (option 155)	80
7.1.3	Machining gear teeth (option 157)	80
7.1.4	Advanced turning functions (option 158)	80
7.2	New functions	80
7.2.1	NC fundamentals	81
7.2.2	Machining cycles	81
7.2.3	Touch probe cycles	81
7.2.4	Program functions	82
7.2.5	Programming Q parameters	82
7.2.6	Programming aids	82
7.2.7	Test run	82
7.2.8	File management	82
7.2.9	CAD Viewer	83
7.2.10	Batch Process Manager (option 154)	83
7.2.11	Program run	83
7.2.12	Global Program Settings (option 44)	83
7.2.13	Positioning w/ Manual Data Input operating mode	83
7.2.14	Status displays	83
7.2.15	Manual touch probe cycles	84
7.2.16	Tilting the working plane (option 8)	84
7.2.17	Tables	84
7.2.18	Tools	84
7.2.19	MOD functions	84
7.2.20	HEROS functions	84
7.2.21	Machine-specific user parameters	85
7.3	Modified functions	85
7.3.1	NC fundamentals	85
7.3.2	Machining cycles	85
7.3.3	Touch probe cycles	86
7.3.4	Program functions	86
7.3.5	Programming Q parameters	86
7.3.6	Test run	86
7.3.7	CAD Viewer	87
7.3.8	Dynamic Collision Monitoring (DCM) (option 40)	87
7.3.9	Moving the machine axes	87
7.3.10	Manual touch probe cycles	87
7.3.11	Tables	87
7.3.12	Tools	87
7.3.13	Remote Desktop Manager (option 133)	87
7.3.14	HEROS functions	87

8	Software 34059x-10	89
8.1	New software options	90
8.1.1	OPC UA NC Server (options 56 to 61)	90
8.1.2	Grinding operations on milling machines (option 156)	90
8.1.3	Optimized contour cycles (option 167)	91
8.1.4	Display step	91
8.1.5	KinematicsComp (option 52)	91
8.2	New functions	91
8.2.1	Machining cycles	92
8.2.2	Program functions	93
8.2.3	Miscellaneous functions	93
8.2.4	Programming Q parameters	94
8.2.5	Calling an external NC program	94
8.2.6	Test run	94
8.2.7	Batch Process Manager (option 154)	94
8.2.8	Program run	94
8.2.9	Global Program Settings (option 44)	95
8.2.10	Status displays	95
8.2.11	Moving the machine axes	95
8.2.12	Tilting the working plane (option 8)	95
8.2.13	Tables	95
8.2.14	Tools	95
8.2.15	Error messages and help system	96
8.2.16	HEROS functions	96
8.2.17	Machine-specific user parameters	96
8.3	Modified functions	96
8.3.1	Machining cycles	97
8.3.2	Touch probe cycles	97
8.3.3	Program functions	97
8.3.4	Adaptive Feed Control (AFC) (option 45)	97
8.3.5	Miscellaneous functions	97
8.3.6	Programming Q parameters	98
8.3.7	Test run	98
8.3.8	File management	98
8.3.9	Program run	98
8.3.10	Status displays	98
8.3.11	Moving the machine axes	98
8.3.12	Manual touch probe cycles	99
8.3.13	Tables	99
8.3.14	Tools	99
8.3.15	MOD functions	99
8.3.16	HEROS functions	99

9 Software 34059x-11.....	101
9.1 New software options.....	102
9.1.1 Renamed software options.....	102
9.2 New functions.....	102
9.2.1 NC fundamentals.....	102
9.2.2 Machining cycles.....	102
9.2.3 Touch probe cycles.....	103
9.2.4 Program functions.....	103
9.2.5 Programming Q parameters.....	104
9.2.6 Programming aids.....	104
9.2.7 Test run.....	104
9.2.8 File management.....	104
9.2.9 CAD Viewer.....	104
9.2.10 Dynamic Collision Monitoring (DCM) (option 40).....	105
9.2.11 Status displays.....	105
9.2.12 Tools.....	105
9.2.13 Error messages and help system.....	105
9.2.14 Accessories.....	105
9.2.15 MOD functions.....	105
9.2.16 Remote Desktop Manager (option 133).....	106
9.2.17 HEROS functions.....	106
9.2.18 Machine-specific user parameters.....	106

9.3	Modified functions.....	106
9.3.1	NC fundamentals.....	106
9.3.2	Machining cycles.....	107
9.3.3	Touch probe cycles.....	108
9.3.4	Miscellaneous functions.....	108
9.3.5	Programming Q parameters.....	108
9.3.6	Programming aids.....	108
9.3.7	Test run.....	109
9.3.8	CAD Viewer.....	109
9.3.9	Batch Process Manager (option 154).....	109
9.3.10	Program run.....	109
9.3.11	Pallets.....	109
9.3.12	Status displays.....	109
9.3.13	Moving the machine axes.....	109
9.3.14	Manual touch probe cycles.....	109
9.3.15	Tables.....	109
9.3.16	Tools.....	110
9.3.17	Error messages and help system.....	110
9.3.18	MOD functions.....	110
9.3.19	Remote Desktop Manager (option 133).....	110
9.3.20	HEROS functions.....	110
9.3.21	Machine-specific user parameters.....	110

10 Software 34059x-16.....	111
10.1 New software options.....	112
10.1.1 CAD Model Optimizer (option 152).....	112
10.1.2 Camera-based inspection of the setup situation (VSC).....	112
10.1.3 Production data acquisition.....	112
10.2 New functions.....	112
10.2.1 NC fundamentals.....	112
10.2.2 Machining cycles.....	113
10.2.3 Touch probe cycles.....	113
10.2.4 Program functions.....	114
10.2.5 Programming Q parameters.....	114
10.2.6 Calling an external NC program.....	114
10.2.7 Programming aids.....	114
10.2.8 File management.....	114
10.2.9 CAD Viewer.....	114
10.2.10 Program run.....	115
10.2.11 Pallets.....	115
10.2.12 Status displays.....	115
10.2.13 Tables.....	115
10.2.14 Tools.....	115
10.2.15 Accessories.....	115
10.2.16 Machine-specific user parameters.....	115
10.3 Modified functions.....	115
10.3.1 NC fundamentals.....	116
10.3.2 Machining cycles.....	116
10.3.3 Touch probe cycles.....	118
10.3.4 Program functions.....	118
10.3.5 Programming Q parameters.....	118
10.3.6 Programming aids.....	119
10.3.7 File management.....	119
10.3.8 CAD Viewer.....	119
10.3.9 Manual touch probe cycles.....	119
10.3.10 Tools.....	119
10.3.11 Error messages and help system.....	119
10.3.12 HEROS functions.....	119
10.3.13 Machine-specific user parameters.....	119

11 Software 34059x-17.....	121
11.1 New software options.....	122
11.1.1 Advanced turning functions.....	122
11.2 New functions.....	122
11.2.1 Touch probe cycles.....	122
11.2.2 Programming Q parameters.....	122
11.2.3 Tables.....	123
11.2.4 Tools.....	123
11.2.5 MOD functions.....	123
11.2.6 HEROS functions.....	123
11.3 Modified functions.....	123
11.3.1 Machining cycles.....	124
11.3.2 Touch probe cycles.....	124
11.3.3 Program functions.....	124
11.3.4 Test run.....	124
11.3.5 CAD Viewer.....	125
11.3.6 Monitoring of configured machine components for overload (option 155).....	125
11.3.7 Moving the machine axes.....	125
11.3.8 Tables.....	125
11.3.9 Tools.....	125
11.3.10 Accessories.....	125
11.3.11 HEROS functions.....	125
11.3.12 Machine-specific user parameters.....	126

1

Software 34059x-02

1.1 New software options

Further information: User's Manual for HEIDENHAIN Conversational Programming (ID: 892903-11)

1.1.1 Dynamic Collision Monitoring (DCM) (option 40)

The machine manufacturer can define machine components and minimum distances that are to be monitored by the control during all machine movements. If two objects monitored for collision come within a defined minimum distance of each other, the control generates an error message and terminates the movement. You can then traverse the machine axes by means of the axis keys only in a direction increasing the distance.

In **Manual Operation** mode, you can deactivate **DCM** individually for program run and manual operation. The general status display shows the current **DCM** status by means of an icon.

1.1.2 CAD Import (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.

1.1.3 Adaptive Feed Control (AFC) (option 45)

AFC controls the contouring feed rate based on the spindle power. After a teach-in cut, **AFC** automatically adapts the feed rate to the varying spindle load (e.g., by reducing the amount of material to be removed). In addition, the control can inspect the tools for wear or breakage.

You define the **AFC** settings in the table AFC.tab and select them in the **AFC** column of the tool table. The control displays the current values on the **AFC** tab of the additional status display. You activate the **AFC** function by soft key during program run. When **AFC** is active, a corresponding icon is shown in the general status display.

1.1.4 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).

1.1.5 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**).

1.1.6 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.

1.1.7 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.

1.1.8 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.

1.1.9 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: 892903-11)

Further information: User's Manual for Cycle Programming (ID: 892905-11)

1.2.1 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.

- Cycle **841 SIMPLE REC. TURNG., RADIAL DIR.** (ISO: **G841**) and Cycle **851 SIMPLE REC TURNG, AX** (ISO: **G851**, option 50)

These cycles enable the recess turning of right-angled slots on the cylindrical surface or front face. You define the slot in the cycles. You can use Cycle **841 (G841)** for both outside and inside machining.

- Cycle **842 ENH.REC.TURNNG, RAD.** (ISO: **G842**) and Cycle **852 ENH.REC.TURNING, AX.** (ISO: **G852**, option 50)

These cycles enable the recess turning of slots on the cylindrical surface or front face. Compared with Cycles **841 (G841)** and **851 (G851)**, these cycles additionally allow you to use:

- Radius or chamfer as a starting or end element
- Inclined slot walls
- Rounding of inside corners

- Cycle **840 RECESS TURNG, RADIAL** (ISO: **G840**) and Cycle **850 RECESS TURNG, AXIAL** (ISO: **G850**, option 50)

These cycles enable the recess turning of slots of any contour. You define the slot contour in a subprogram and select the contour using Cycle **14 CONTOUR** (ISO: **G37**).

- The turning cycles (option 50) are available in ISO programming mode.

1.2.2 Touch probe cycles

- 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 cycle 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**)

1.2.3 Miscellaneous functions

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

1.2.4 Program functions

- **FUNCTION TURNDATA BLANK** (option 50) allows you to program the updating of the blank form for turning operations. The control automatically detects any residual material and moves the tool along the shortest path without air cuts. You define the workpiece blank as a closed contour in a subprogram or an external NC program.
- 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.

1.2.5 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.

1.2.6 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.

1.2.7 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. The machine manufacturer can also permanently assign the virtual tool axis to an axis key on the handwheel.

1.2.8 Manual touch probe cycles

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

1.2.9 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.

1.2.10 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.

1.2.11 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: 892903-11)

Further information: User's Manual for Cycle Programming (ID: 892905-11)

1.3.1 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 are retained.

1.3.2 Machining cycles

- The parameter **Q437 APPROACH POSITION** has been added to Cycle **256 RECTANGULAR STUD** (ISO: **G256**). 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**). 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**) supports the **APPR** and **DEP** functions in the contour description.
- You can call the SL Cycles even if radius compensation is active.
- The parameter **Q463 LIMIT TO DEPTH** that allows you to define the maximum plunging depth per cut has been added to all of the recessing cycles (option 50).

1.3.3 Touch probe cycles

- 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**.

1.3.4 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.
- In **FUNCTION TURNDATA SPIN** (option 50), you can use the parameter **S MAX** to program a maximum spindle speed.
- In **FUNCTION TURNDATA CORR** (option 50), you define the compensation value first in **Z** and then in **X**. The sequence is compatible with the definition of a milling cutter.

1.3.5 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.

1.3.6 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.

1.3.7 Program run

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

1.3.8 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.

1.3.9 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.

1.3.10 Moving the machine axes

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

1.3.11 Manual touch probe cycles

- 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.
- The control also supports the manual touch probe cycles during turning mode (option 50).
- 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.

1.3.12 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**.

1.3.13 Tools

- The maximum number of characters permitted in the **NAME** and **DOC** columns of the tool table has been increased from 16 to 32.
- The tool table provides a soft key that allows you to switch to the turning tool table (option 50).
- If, in the form view of the tool management, you place the cursor into the **TYPE** field and press the **GOTO** key, then the control displays help graphics for the individual tool types.
- The **+** and **-** keys allow you to activate and deactivate check boxes in the form view of the tool management.
- The **UPDATE** soft key in the tool management has been renamed as **UPDATE THE VIEW**.
- When you delete tools or import or export tool data in the tool management, the control highlights the tool currently being edited in blue.

1.3.14 Machine-specific user parameters

When you edit an NC block and press the up or down cursor key, the control goes to the next identical syntax element. The machine parameter **maxLineCommandSrch** (no. 105412) defines the number of lines that are searched by the control. The input range of this parameter has been increased from 10 000 lines to 100 000 lines. The default value has been changed from 500 lines to 10 000 lines.

2

Software 34059x-04

2.1 New software options

Further information: User's Manual for HEIDENHAIN Conversational Programming (ID: 892903-12)

2.1.1 Synchronizing functions (option 135)

RTC (Real Time Coupling) enables complex, simultaneous machine movements through enhanced synchronization of NC axes, auxiliary axes, and spindles.

You can use Cycle **800 ADJUST XZ SYSTEM** (ISO: **G800**, option 50) for eccentric turning.

Further information: "Machining cycles", Page 32

2.1.2 Active Vibration Damping (AVD) (option 146)

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

2.1.3 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: 892903-12)

Further information: User's Manual for Cycle Programming (ID: 892905-12)

2.2.1 NC fundamentals

- The control supports up to four spindles.
- 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.

2.2.2 Machining cycles

- Cycle **233 FACE MILLING** (ISO: **G233**)

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**)

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.

2.2.3 Touch probe cycles

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.

2.2.4 Program functions

- **FUNCTION TURNDATA CORR-TCS** (option 50) allows you to compensate a turning tool in the tool coordinate system. This compensation is also effective during inclined machining.
- **FUNCTION TURNDATA CORR-WPL** (option 50) allows you to compensate a turning tool in the workpiece coordinate system. This compensation has the same effect as a workpiece oversize.
- If you switch the machining mode using **FUNCTION MODE** (option 50), you can choose a kinematic model based on your machine model.

2.2.5 Adaptive Feed Control (AFC) (option 45)

The following functions allow you to regulate **AFC** from within the NC program:

- **FUNCTION AFC CUT BEGIN** starts an **AFC** cutting sequence. The control changes from the teach-in cut to feedback control mode as soon as a reference power has been determined or one of the conditions **TIME**, **DIST**, or **LOAD** has been met. The parameters **TIME** and **DIST** define a maximum time and the cutting length of the teach-in cut, respectively, while **LOAD** defines a reference load.
- **FUNCTION AFC CUT END** allows you to deactivate **AFC**.
- **FUNCTION AFC CTRL** immediately starts the feedback control mode, even if the teach-in cut has not yet been terminated.

2.2.6 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
 - **FN 18: SYSREAD (D18) ID310 NR20**: check for whether the coordinates are referenced to the diameter or radius during turning

2.2.7 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.

2.2.8 Test run

- 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. In addition, the simulation displays indexable inserts.

2.2.9 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.

2.2.10 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.

2.2.11 Tables

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

2.2.12 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.
- The **NAME** column has been added to the turning tool table (option 50). The control automatically loads the name from the tool table.

2.2.13 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: 892903-12)

Further information: User's Manual for Cycle Programming (ID: 892905-12)

2.3.1 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.

2.3.2 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**), 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**). 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**). 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**) 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.
- 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**). 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**) check for whether the actual pitch matches the pitch defined in the **PITCH** column of the tool table.
- The parameter **Q499 REVERSE CONTOUR** has been added to Cycles **840 RECESS TURNG, RADIAL** (ISO: **G840**) and **850 RECESS TURNG, AXIAL** (ISO: **G850**, option 50). This parameter allows you to machine the contour in the direction opposite to the programmed direction.
- Cycle **800 ADJUST XZ SYSTEM** (ISO: **G800**, option 50) has been enhanced. It remains active until it is reset by Cycle **801 RESET ROTARY COORDINATE SYSTEM** (ISO: **G801**) or a new Cycle **800 (G800)** is programmed.
You can use this cycle for inclined machining or eccentric turning (option 135).
The following parameters have been added:
 - **Q530 INCLINED MACHINING**: positioning behavior
 - **Q531 ANGLE OF INCIDENCE**: angle of inclined machining operation
 - **Q532 FEED RATE**: feed rate of tilting movement
 - **Q533 PREFERRED DIRECTION**: tilting in the positive or negative direction
 - **Q535 ECCENTRIC TURNING**: coupling of axes with the lathe spindle (option 135)
 - **Q536 ECCENTRIC W/O STOP**: suppressing the additional confirmation (option 135)

2.3.3 Touch probe cycles

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.

2.3.4 Miscellaneous functions

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

2.3.5 Programming Q parameters

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

2.3.6 Programming aids

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

2.3.7 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.

2.3.8 Dynamic Collision Monitoring (DCM) (option 40)

DCM monitors movements, in which the machine traverses more than one axis (e.g., movements with active **FUNCTION TCPM** (option 9)).

2.3.9 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.

2.3.10 Manual touch probe cycles

- 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.

2.3.11 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.

2.3.12 Tools

During pallet machining, the control highlights each main program in color on the **T usage order** tab of the extended tool management (option 93).

2.3.13 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.

2.3.14 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.

2.3.15 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 100 000 lines to 200 000 lines. The default value has been changed from 10 000 lines to 20 000 lines.

3

Software 34059x-05

3.1 New software options

Further information: User's Manual for HEIDENHAIN Conversational Programming (ID: 892903-13)

Further information: User's Manual for Cycle Programming (ID: 892905-13)

3.1.1 KinematicsComp (option 52)

KinematicsComp enables spatial compensation for the errors of the linear and rotary axes. This allows the control to compensate for position and component errors. The machine manufacturer configures the option.

This option requires an export permit.

3.1.2 Advanced Spindle Interpolation (option 96)

This software option provides the interpolation turning function that enables turning operations without the need for option 50.

The option includes the following cycles:

- Cycle **291 COUPLG.TURNG.INTERP.** (ISO: **G291**)
- Cycle **292 CONTOUR.TURNG.INTRP.** (ISO: **G292**)

Further information: "Machining cycles", Page 37

3.1.3 Spindle Synchronism (option 131)

This option allows you to rotate two or more spindles synchronously, with a gear ratio, or a defined offset.

The option includes the following cycle:

- Cycle **880 GEAR HOBBING** (ISO: **G880**, option 50)

Further information: "Machining cycles", Page 37

3.1.4 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**

3.2 New functions

Further information: User's Manual for HEIDENHAIN Conversational Programming (ID: 892903-13)

Further information: User's Manual for Cycle Programming (ID: 892905-13)

3.2.1 NC fundamentals

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

3.2.2 Machining cycles

- Cycle **270 CONTOUR TRAIN DATA** (ISO: **G270**)
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)
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 **291 COUPLG.TURNG.INTERP.** (ISO: **G291**, option 96)
This cycles allows the control to couple the tool spindle with the positions of the linear axes. This cycle allows you to execute a turning operation without the need for option 50. When the cycle is active, the control aligns the tool tip to the center of a defined circle.
- Cycle **292 CONTOUR.TURNG.INTRP.** (ISO: **G292**, option 96)
This cycle allows you to finish a rotationally symmetric contour without the need for option 50. The cycle couples the tool spindle with the positions of the linear axes.
You define the contour in a subprogram and select it with Cycle **14 CONTOUR** (ISO: **G37**) or the function **SEL CONTOUR** (ISO: **:%:CNT**). You can machine the contour with a turning tool or a milling cutter.
- Cycle **880 GEAR HOBBING** (ISO: **G880**, option 50, option 131)
This cycle allows you to machine cylindrical gears with straight or oblique external gear teeth. During machining, the control couples the tool spindle with the rotary table. You define the gear wheel and the tool in the cycle parameters.
- 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.

3.2.3 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.

3.2.4 Dynamic Collision Monitoring (DCM) (option 40)

You can control **DCM** from within the NC program. **FUNCTION DCM OFF** allows you to temporarily deactivate **DCM** until **FUNCTION DCM ON** is programmed or the NC program ends.

3.2.5 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.

3.2.6 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 NRO**: 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

3.2.7 File management

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

3.2.8 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.

3.2.9 Manual touch probe cycles

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.

3.2.10 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.

3.2.11 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**
- In the column **YL** of the turning tool table (option 50), you define the tool length in the Y direction.
- In the column **DYL** of the turning tool table (option 50), you define the tool length oversize in the Y direction.
- For **Test Run** operating mode, you can select any turning tool table (option 50) that contains the same turning tools as the active tool table. The selected table is identified with the status "**S**" in the file manager.

3.2.12 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.

3.2.13 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).

3.2.14 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: 892903-13)

Further information: User's Manual for Cycle Programming (ID: 892905-13)

3.3.1 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**).
- The pop-up window for tool selection displays the tool type. For a turning tool (option 50), the pop-up window shows the values from the columns **ZL** and **XL** of the turning tool table.
- 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**

3.3.2 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**) 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**) 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**), then the control also considers the parameter **Q370 TOOL PATH OVERLAP** during the first infeed.
- Cycle **22 ROUGH-OUT** (ISO: **G122**) 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
- The parameter **Q488 PLUNGING FEED RATE** has been added to Cycles **841 SIMPLE REC. TURNG., RADIAL DIR.** (ISO: **G841**), **842 ENH.REC.TURNNG, RAD.** (ISO: **G842**), **851 SIMPLE REC TURNG, AX** (ISO: **G851**), and **852 ENH.REC.TURNING, AX.** (ISO: **G852**, option 50). This parameter allows you to define a feed rate for the infeed.
- Cycle **800 ADJUST XZ SYSTEM** (ISO: **G800**, option 50) enables eccentric turning without the need for option 135.

3.3.3 Touch probe cycles

- 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.

3.3.4 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).

3.3.5 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.

3.3.6 Programming aids

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

3.3.7 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.

3.3.8 DXF Converter (option 42)

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

3.3.9 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.

3.3.10 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.

3.3.11 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.

3.3.12 Manual touch probe cycles

- 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.

3.3.13 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.

3.3.14 Tools

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

3.3.15 Machine-specific user parameters

- The input range of the machine parameter **maxLineCommandSrch** (no. 105412) has been increased from 200 000 lines to 400 000 lines.
- 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 100 000 lines. The default value has been changed from 100 lines to 20 000 lines.

4

Software 34059x-06

4.1 New software options

Further information: User's Manual for Conversational Programming (ID: 892903-14)

Further information: User's Manual for Cycle Programming (ID: 892905-14)

4.1.1 Camera-based Visual Setup Control (VSC) (option 136)

VSC enables visual control of the setup situation of a workpiece by means of a camera. You take reference images corresponding to the nominal condition.

During machining, the controls compares the actual condition of the monitoring area with the reference images. You define how the control responds to an error. The Q parameter **Q601** shows the current error status.

The function provides the following cycles:

- Cycle **600 GLOBAL WORKING SPACE** (ISO: **G600**)
- Cycle **601 LOCAL WORKING SPACE** (ISO: **G601**)

Further information: "Touch probe cycles", Page 46

4.2 New functions

Further information: User's Manual for Conversational Programming (ID: 892903-14)

Further information: User's Manual for Cycle Programming (ID: 892905-14)

4.2.1 Machining cycles

Cycle **258 POLYGON STUD** (ISO: **G258**)

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

4.2.2 Touch probe cycles

- Cycle **600 GLOBAL WORKING SPACE** (ISO: **G600**, option 136)
This cycle allows you to take an image of the working space at a position defined by the machine manufacturer.
- Cycle **601 LOCAL WORKING SPACE** (ISO: **G601**, option 136)
This cycle allows you to take an image of the working space at the current position.

4.2.3 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.

4.2.4 Adaptive Feed Control (AFC) (option 45)

In the **AFC-LOAD** column of the tool table, you can define a spindle reference power for **AFC** in percent, which you determine once by means of a teach-in cut. If a reference power is defined, the control will not perform a teach-in cut.

4.2.5 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

4.2.6 Programming aids

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

4.2.7 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.

4.2.8 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.

4.2.9 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.

4.2.10 Pallets

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

4.2.11 Moving the machine axes

The machine manufacturer can configure the workpiece spindle (option 50) as an axis that can be selected on the handwheel.

4.2.12 Manual touch probe cycles

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.

4.2.13 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.

4.2.14 Tools

- The control takes into account tool carriers. The additional **ToolHolderWizard** tool allows you to create tool carriers or parameterize available 3-D models. The control uses **DCM** (option 40) to monitor the tool carriers.
- In the **KINEMATIC** column of the tool table, you use the **SELECT** soft key to assign a tool-carrier kinematic model to a tool.

4.2.15 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**).

4.3 Modified functions

Further information: User's Manual for Conversational Programming (ID: 892903-14)

Further information: User's Manual for Cycle Programming (ID: 892905-14)

4.3.1 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 provides the **START** and **START SINGLE** soft keys. On a 19-inch screen, the soft-key row also provides the **RESET + START** soft key.

4.3.2 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**), 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**)
 - Cycle **233 FACE MILLING** (ISO: **G233**)
 - Cycle **292 CONTOUR.TURNG.INTRP.** (ISO: **G292**, option 96)
- The SL Cycles 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**), 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**), 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**), 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.
- The parameter **Q561 CONVERT FROM TURNING TOOL** has been added to Cycle **291 COUPLG.TURNG.INTERP.** (ISO: **G291**, option 96). If **Q561 = 1**, then the control interprets the value of column **XL** of the turning tool table in the same way as the radius **R** for milling cutters and enables programming with radius compensation.
- The input options of +2 and -2 have been added to the parameter **Q533 PREFERRED DIRECTION** in Cycles **800 ADJUST XZ SYSTEM** (ISO: **G800**, option 50) and **880 GEAR HOBBING** (ISO: **G880**, option 50, option 131). These input options allow you to define that the angle is tilted in the positive or negative direction within a range from 90° to 180°.

4.3.3 Touch probe cycles

- 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 (option 50).
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.

4.3.4 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.

4.3.5 DXF Converter (option 42)

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

4.3.6 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.

4.3.7 Pallets

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.

4.3.8 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.

4.3.9 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.

4.3.10 Manual touch probe cycles

- 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.

4.3.11 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**.

4.3.12 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.

4.3.13 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 34059x-07

5.1 New software options

Further information: User's Manual for Conversational Programming (ID: 892903-15)

Further information: User's Manual for Cycle Programming (ID: 892905-15)

5.1.1 3-D radius compensation depending on the tool's contact angle (option 92)

This option allows you to compensate for tools depending on the tool contact angle. For this purpose, you save the form inaccuracy of a touch probe or a milling cutter with its cutting radius to a compensation value table. The control considers the compensation values in NC programs with surface normal vectors.

You determine the compensation values for a touch probe by using Cycle **460 CALIBRATION OF TS ON A SPHERE** (ISO: **G460**). During this cycle, the control probes multiple points along a quarter circle and saves the deflection behavior of the touch probe at the various approach angles. You determine the compensation values for milling cutters by means of a laser, for example.

In the **DR2TABLE** column of the tool table, you assign a compensation-value table to a tool.

The control considers the compensation values in Cycle **444 PROBING IN 3-D** (ISO: **G444**).

This option requires an export permit.

5.2 New functions

Further information: User's Manual for **Klartext Programming** - 892903-15

Further information: User's Manual for **Programming of Machining Cycles** - 892905-15

5.2.1 NC fundamentals

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

5.2.2 Touch probe cycles

Cycle **444 PROBING IN 3-D** (ISO: **G444**)

This cycle allows the control to probe a point that is defined in three axes and compare the current position with the nominal position. You define a normal vector on which the tolerance range is based.

The control saves the measured values and deviations to Q parameters and outputs a measuring log in HTML format.

5.2.3 Program functions

- **FUNCTION DWELL** allows you to program a dwell time in seconds or define the number of spindle revolutions for dwelling.
- **FUNCTION TURNDATA CORR-TCS** allows you to define an oversize for the width of a recessing tool by using the parameter **DCW** (option 50).

5.2.4 Adaptive Feed Control (AFC) (option 45)

AFC enables the monitoring of the tool for wear and breakage during machining. **AFC** uses the current spindle power for monitoring and responds based on defined limit values.

You define the limit values for the early warning level and the switch-off level in the **AFC-OVLD1** and **AFC-OVLD2** columns of the tool table.

5.2.5 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) ID951**: data of the current turning tool from the turning tool table
 - **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.

5.2.6 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.

5.2.7 Test run

- 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.
- The **GENERATE TOOL USAGE FILE** soft key allows you to create a tool usage file without having to simulate the NC program.

5.2.8 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**.

5.2.9 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.

5.2.10 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.

5.2.11 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.

5.2.12 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 .

5.2.13 Tilting the working plane (option 8)

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

5.2.14 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.
- In the column **DCW** of the turning tool table (option 50), you define an oversize for the width of a recessing tool.
- The control saves the active tool length of a turning tool (option 50) to the Q parameter **Q114**.
- In the tool management, you can manually correct the measured compensation values of a turning tool (option 50). In the form view, the **CALCULATE TOOL COMPENSTN.** soft key allows you to open a dialog for calculating the compensation. The control converts the entered data into the tool coordinate system.

5.2.15 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

5.2.16 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.

5.2.17 Machine-specific user parameters

- During many face turning operations, it is not possible to maintain constant surface speeds because the maximum spindle speed **S_{MAX}** is reached before the center of rotation. The optional machine parameter **facMinFeedTurnS_{MAX}** (no. 201009) allows you to define whether the feed rate is reduced after the maximum spindle speed has been reached. You define a factor of the programmed feed rate, which then becomes effective as the minimum feed rate.
- 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**.

5.3 Modified functions

Further information: User's Manual for **Klartext Programming** - 892903-15

Further information: User's Manual for **Programming of Machining Cycles** - 892905-15

5.3.1 NC fundamentals

- Cylindrical workpiece blanks can also be defined by means of a diameter.
- You can program up to six axes in a linear block.
- 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.

5.3.2 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**) has been changed from 0.1 to 0.001.
- Path generation has been optimized in Cycle **233 FACE MILLING** (ISO: **G233**). The number of lateral infeeds has been reduced.
- The parameter **Q439 FEED RATE REFERENCE** has been added to Cycle **251 RECTANGULAR POCKET** (ISO: **G251**). 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**).
- The parameter **Q438 ROUGH-OUT TOOL** has been added to Cycle **24 SIDE FINISHING** (ISO: **G124**). This parameter allows you define the tool for roughing out the contour.
- Cycle **275 TROCHOIDAL SLOT** (ISO: **G275**) 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.
- The input option 2 has been added to the parameter **Q499 REVERSE CONTOUR** in Cycles **810 TURN CONTOUR LONG.** (ISO: **G810**), **815 CONTOUR-PAR. TURNING** (ISO: **G815**), and **820 TURN CONTOUR TRANSV.** (ISO: **G820**, option 50). If you select this option, the control reverses the contour and changes the radius compensation.
- The recessing cycles (option 50) have been enhanced in order to increase the tool life:
 - The parameter **Q510 RECESSING OVERLAP** allows you to define an overlap factor for stepover. The control calculates the stepover from this factor and the width of the tool.
 - The parameter **Q511 FEED RATE FACTOR** allows you to define a percentage factor for the feed rate defined in the parameter **Q478 ROUGHING FEED RATE** for full-section cutting (e.g., for the first cut of each infeed).
 - The parameter **Q462 RETRACTION MODE** allows you to choose whether the control retracts the tool along the contour after a cut or first departs the contour along a diagonal path.

5.3.3 Touch probe cycles

- 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.
- In conjunction with option 92, you can use Cycle **460 CALIBRATION OF TS ON A SPHERE** (ISO: **G460**) for 3-D calibration of a touch probe. For this purpose, the parameter **Q455 NO. POINTS 3-D CAL.** has been added.
- The input option 2 has been added to the parameter **Q406 MODE** in Cycle **451 MEASURE KINEMATICS** (ISO: **G451**, option 48). When option 52 is active, this input option allows the control to compensate for the angular and position errors of the machine.
- 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.
- The responses of Cycles **600 GLOBAL WORKING SPACE** (ISO: **G600**) and **601 LOCAL WORKING SPACE** (ISO: **G601**, option 136) have been modified.
The control issues an error message for the following scenarios:
 - No reference image defined for the monitoring point in parameter **QS600**
 - No monitoring range and no error defined for monitoring point in parameter **QS600**

5.3.4 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**.

5.3.5 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.

5.3.6 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.

5.3.7 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

5.3.8 Test run

- 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.

5.3.9 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.

5.3.10 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.

5.3.11 Dynamic Collision Monitoring (DCM) (option 40)

The machine manufacturer defines the response of the control if a collision occurs due to a retraction that has been defined with **M140**. The control either stops machining or terminates the retraction movement ahead of time and continues the NC program.

5.3.12 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 by means of a block scan works properly.
- The control issues an error message when a touch probe cycle is run within the NC program without a touch probe being inserted.

5.3.13 Pallets

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

5.3.14 Status displays

- When a spindle speed limit defined with **FUNCTION TURNDATA SPIN SMAX** (option 50) is active, then the control displays **SMAX** instead of **S** in the status display.
- 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.

5.3.15 Camera-based monitoring of setup situation (VSC) (option 136)

- If you press the **SHOW AREAS** soft key, the control shows all monitoring areas of an image.
- If you press the **SHOW COMPARISON** soft key, the control compares the current image with the mean image.
- The **VSC SETTINGS** soft key allows the machine manufacturer to configure the field of view of the camera.
- If an image contains several monitoring or error areas, only one area is active at any one time and can be edited or deleted. You select the active area with the mouse.

5.3.16 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.

5.3.17 Manual touch probe cycles

- 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).

5.3.18 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.

5.3.19 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.

5.3.20 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.
- The soft key for opening the turning tool table (option 50) has been renamed **TURNING TOOLS** from **TOOL TABLE**.

5.3.21 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.

5.3.22 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 34059x-08

6.1 New software options

Further information: User's Manual for Conversational Programming (ID: 892903-16)

6.1.1 Global Program Settings (option 44)

This option allows you to define various coordinate transformations and settings for the operating modes **Program run, single block**, **Program run, full sequence**, and **Positioning w/ Manual Data Input**. These settings are globally effective in addition to the settings in the NC program.

The **GLOBAL SETTINGS** soft key allows you to open a form in which you define the settings options enabled by the machine manufacturer.

The **Global Program Settings** comprises the following setting options:

- **Additive offset (M-CS)**
- **Additive basic rotat. (W-CS)**
- **Shift (W-CS)**
- **Mirroring (W-CS)**
- **Shift (mW-CS)**
- **Rotation (WPL-CS)**
- **Handwheel superimp.**
- **Coordinate system**
- **Feed rate factor**

The **GS** and **POS HR** tabs have been added to the additional status display. The **GS** tab contains the current values of all settings options and the **POS HR** tab shows the current **Handwheel superimp.** values. When the **Global Program Settings** function is active, the control displays a corresponding icon in the general status display.

6.1.2 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 .

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

6.1.3 Renamed software options

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

6.2 New functions

Further information: User's Manual for Conversational Programming (ID: 892903-16)

Further information: User's Manual for Cycle Programming (ID: 892905-16)

6.2.1 NC fundamentals

The control supports up to 24 control loops, of which up to four are configurable as spindles.

6.2.2 Machining cycles

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

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.

6.2.3 Touch probe cycles

- 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.

- Cycle **453 KINEMATICS GRID** (ISO: **G453**, option 48, option 52)

This cycle enables the control to probe a calibration sphere in multiple rotary axis positions defined by the machine manufacturer. The control ascertains residual position errors at the tool center point (**TCP**) that are still present even after optimization of the kinematics by means of Cycle **451 MEASURE KINEMATICS** (ISO: **G451**). The machine must be prepared by the machine manufacturer for this cycle.

6.2.4 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 **FACING HEAD POS** (option 50) function enables machining operations with a facing head. This function allows the control to convert the values programmed for the X axis into values for the U axis of the facing head. For this to work, you need to select a suitable kinematic model for a facing head in **FUNCTION MODE TURN**.
- **FUNCTION TURNDATA CORR-TCS** allows you to define an allowance for the cutting radius of a turning tool by using the parameter **DRS** (option 50).
- 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.

6.2.5 Adaptive Feed Control (AFC) (option 45)

In turning mode (option 50), **AFC** is available with a reduced performance range. During machining the control monitors the tool for wear and breakage, but it does not adapt the feed rate. The entire learning phase must be completed because the values cannot be predefined.

The following columns of the table AFC.tab need to be defined:

- **OVLD**: overload response
- **POUT**: minimum load
- **SENS**: sensitivity
- **PLC**: PLC setting

In the **AFC** column of the tool table, you select the monitoring settings from the table AFC.tab.

6.2.6 Miscellaneous functions

M138 allows you to select individual rotary axes for tilting, **M128**, or **FUNCTION TCPM** (option 9) even in turning mode (option 50).

6.2.7 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.

6.2.8 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.

6.2.9 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.

6.2.10 Test run

- 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.
- You can activate **DCM** (option 40) by soft key in **Test Run** operating mode. If **DCM** detects a collision during simulation, the control stops simulation and displays the colliding objects.

6.2.11 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.

6.2.12 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.

6.2.13 Pallets

- 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.

6.2.14 Status displays

- When mirroring is active, the control displays a corresponding icon in the general status display.
- When **AFC** (option 45) is active in teach-in cut mode, the control displays a corresponding icon in the general status display.
- For turning tools (option 50), the **TOOL** tab of the additional status display contains the tool orientation **TO**.

6.2.15 Manual touch probe cycles

- 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.

6.2.16 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. 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).

6.2.17 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.
- The **DRS** column for defining an allowance for the cutting radius has been added to the turning tool table (option 50).

6.2.18 Accessories

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

6.2.19 MOD functions

- The **Counter settings** group that allows you to change the current count and the target value has been added to the MOD menu.
- 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.

6.2.20 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.

6.2.21 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: 892903-16)

Further information: User's Manual for Cycle Programming (ID: 892905-16)

6.3.1 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.

6.3.2 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**) now also provide roughing and finishing operations.
The following parameters have been added:
 - **Q215 MACHINING OPERATION**
 - **Q369 ALLOWANCE FOR FLOOR**
 - **Q338 INFEEED FOR FINISHING**
 - **Q385 FINISHING FEED RATE**
- During finishing in Cycle **233 FACE MILLING** (ISO: **G233**), 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**), 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**) 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**), 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.
- In Cycles **205 UNIVERSAL PECKING** (ISO: **G205**) and **241 SINGLE-LIP D.H.DRLNG** (ISO: **G241**), 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.
- The parameter **Q211 DWELL TIME IN REVS** has been added to the recessing cycles (option 50). This parameter allows you to specify a dwell time in revolutions of the workpiece spindle. This dwell time delays retraction after the recessing operation.
- The input range of the parameter **Q475 NUMBER OF STARTS** has been increased from 255 to 500 in Cycles **830 THREAD CONTOUR-PARALLEL** (ISO: **G830**), **831 THREAD LONGITUDINAL** (ISO: **G831**), and **832 THREAD EXTENDED** (ISO: **G832**, option 50).
- 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 CYCLS** 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.

6.3.3 Touch probe cycles

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.

6.3.4 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 option 50.
 - **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.

6.3.5 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.

6.3.6 Programming Q parameters

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

6.3.7 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.

6.3.8 Test run

- 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.

6.3.9 File management

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

6.3.10 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.
- The control restores a programmed speed limit (option 50) after eccentric turning.

6.3.11 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 **AFC** (option 45) icon in the general status display has been modified so that it shows the character string **AFC**.
- 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.

6.3.12 Camera-based monitoring of setup situation (VSC) (option 136)

When you press the **LIVE IMAGE** soft key in **Manual Operation** mode, then the control opens a pop-up window for selecting the target directory and the file name of the image. The defined path is retained until the control is restarted.

6.3.13 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.

6.3.14 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.

6.3.15 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.

6.3.16 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.

6.3.17 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 34059x-09

7.1 New software options

Further information: User's Manual for **Setup, Testing and Running NC Programs** - 1261174-10

Further information: User's Manual for **Programming of Machining Cycles** - 892905-17

7.1.1 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 production data can still be retrieved even after the corporate network has been down for hours at a time.

7.1.2 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.

7.1.3 Machining gear teeth (option 157)

This software option enables the simple and economical machining of cylindrical gear teeth in one setup.

In Cycle **285 DEFINE GEAR** (ISO: **G285**), you define the gear tooth geometry. You define the tool in the machining cycles. During machining, the control synchronizes the workpiece spindle and the tool spindle and superimposes an axial movement of the tool on the coupled turning movements.

The option includes the following cycles:

- Cycle **285 DEFINE GEAR** (ISO: **G285**)
- Cycle **286 GEAR HOBBING** (ISO: **G286**)
- Cycle **287 GEAR SKIVING** (ISO: **G287**)

Further information: "Machining cycles", Page 81

7.1.4 Advanced turning functions (option 158)

This software option includes Cycle **883 TURNING SIMULTANEOUS FINISHING** (ISO: **G883**) that allows you to finish complex turning contours in one pass.

Further information: "Machining cycles", Page 81

7.2 New functions

Further information: User's Manual for **Klartext Programming** - 892903-17

Further information: User's Manual for **Setup, Testing and Running NC Programs** - 1261174-10

Further information: User's Manual for **Programming of Machining Cycles** - 892905-17

7.2.1 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, 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.

7.2.2 Machining cycles

- Cycle **285 DEFINE GEAR** (ISO: **G285**, option 157)
This cycle allows you to define the geometry of gear teeth. If you define only the parameters **Q540 MODULE** and **Q541 NUMBER OF TEETH**, the control creates gear teeth according to DIN 3960.
- Cycle **286 GEAR HOBGING** (ISO: **G286**, option 157)
This cycle enables the machining of previously defined cylindrical gear teeth with a hobbing process.
- Cycle **287 GEAR SKIVING** (ISO: **G287**, option 157)
This cycle enables the machining of previously defined cylindrical gear teeth with a skiving process.
- Cycle **883 TURNING SIMULTANEOUS FINISHING** (ISO: **G883**, option 50, option 158)
This cycle enables the machining of turning contours using different inclination angles. This allows you to machine complex contours containing undercuts in one pass, for example. In addition, you can increase the service life of the tool by utilizing a large area of the indexable insert.
You define the contour in a subprogram and select it with Cycle **14 CONTOUR** (ISO: **G37**) or the **SEL CONTOUR** function.

7.2.3 Touch probe cycles

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.

7.2.4 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.

7.2.5 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.

7.2.6 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.

7.2.7 Test run

- 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.

7.2.8 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.

7.2.9 CAD Viewer

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

7.2.10 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.

7.2.11 Program run

- When the **MACHINE** screen layout is selected, the control shows the workpiece and the collision objects of the machine. You can also select the screen layouts **PROGRAMM + MACHINE** or **POSITION + MACHINE**.
- You can edit a called NC program as soon as program run has been completed.

7.2.12 Global Program Settings (option 44)

- If you activate **Handwheel superimp.** in the **Global Program Settings** function, the control can show the values of the virtual tool axis **VT** in the position display. You need to define **VT** in the **axisDisplay** machine parameter (no. 100810) for the control to be able to display the virtual tool axis.
- The machine manufacturer can define the axis-specific effect of a **Shift (mW-CS)** of the rotary axes. The machine manufacturer can select the behavior of the iTNC 530 control, in which the setting leads to a change in the tool's angle of inclination.

7.2.13 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.

7.2.14 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
- Values from **DCM** (option 40)

7.2.15 Manual touch probe cycles

- 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

7.2.16 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.

7.2.17 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.

7.2.18 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**

7.2.19 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.

7.2.20 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.

7.2.21 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 **Klartext Programming** - 892903-17

Further information: User's Manual for Setup, Testing and Running NC Programs - 1261174-10

Further information: User's Manual for Cycle Programming - 892905-17

7.3.1 NC fundamentals

- When you program linear blocks, the default is no longer that the control shows the syntax element **&Z**. In order to use **&** to program parallel-axis movements, you press the left cursor key when defining the radius compensation.
- 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, then the control issues an error message.

7.3.2 Machining cycles

- The parameter **Q367 SURFACE POSITION** has been added to Cycle **233 FACE MILLING** (ISO: **G233**). 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**) performs the approach motion and departure motion along a tangential helix, in order to avoid surface blemishes on the floor.
- Cycle **291 COUPLG.TURNG.INTERP.** (ISO: **G291**) and **292 CONTOUR.TURNG.INTRP.** (ISO: **G292**, option 96) take into account the machine parameter **mStrobeOrient** (no. 201005), in order to avoid errors in the spindle coupling.
- The input range of parameter **Q531 ANGLE OF INCIDENCE** has been increased from $\pm 180.0^\circ$ to $\pm 180.000^\circ$ in Cycle **800 ADJUST XZ SYSTEM** (ISO: **G800**, option 50).

7.3.3 Touch probe cycles

- 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 prepositioning.
- 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.
- Cycle **444 PROBING IN 3-D** (ISO: **G444**) takes into account an active **M128** or **FUNCTION TCPM** (option 9).
- Cycle **444 PROBING IN 3-D** (ISO: **G444**) takes into account the machine parameter **chkTiltingAxes** (no. 204601). 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.
- When restoring a kinematic model, Cycle **450 SAVE KINEMATICS** (ISO: **G450**, option 48) changes only those values that differ from the active values.
- An input value of 3 has been added to the parameter **Q406 MODE** in Cycle **451 MEASURE KINEMATICS** (ISO: **G451**, option 48). In this mode, the control compensates the machine datum if option 52 is active.
- 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.

7.3.4 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).

7.3.5 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.

7.3.6 Test run

- 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.
- When a touch probe cycle is simulated in **Test Run** operating mode, the control uses a simulation probe and considers the programmed values.

7.3.7 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.

7.3.8 Dynamic Collision Monitoring (DCM) (option 40)

The machine manufacturer can increase the minimum distance between the collision-monitored objects for **Manual Operation** mode (option 40).

7.3.9 Moving the machine axes

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

7.3.10 Manual touch probe cycles

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.

7.3.11 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.

7.3.12 Tools

- The machine manufacturer can define default values for the **L-OFFS** and **R-OFFS** columns of the tool table.
- You can call the **TNCguide** in the form view of the tool management.

7.3.13 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.

7.3.14 HEROS functions

The screensaver glideshow has been removed.

8

Software 34059x-10

8.1 New software options

Further information: User's Manual for **Klartext Programming** - 892903-18

Further information: User's Manual for **Setup, Testing and Running NC Programs** - 1261174-11

Further information: User's Manual for **Programming of Machining Cycles** - 892905-18

8.1.1 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.

8.1.2 Grinding operations on milling machines (option 156)

This option allows you to perform a grinding operation at the control. The control supports jig grinding that enables the superimposition of a reciprocating stroke onto the path contours.

The option includes the following cycles:

- Cycle **1000 DEFINE RECIP. STROKE** (ISO: **G1000**)
- Cycle **1001 START RECIP. STROKE** (ISO: **G1001**)
- Cycle **1002 STOP RECIP. STROKE** (ISO: **G1002**)
- Cycle **1010 DRESSING DIAMETER** (ISO: **G1010**)
- Cycle **1015 PROFILE DRESSING** (ISO: **G1015**)
- Cycle **1030 ACTIVATE WHEEL EDGE** (ISO: **G1030**)
- Cycle **1032 GRINDING WHL LENGTH COMPENSATION** (ISO: **G1032**)
- Cycle **1033 GRINDING WHL RADIUS COMPENSATION** (ISO: **G1033**)

Further information: "Machining cycles", Page 92

You need to activate **FUNCTION DRESS** to be able to program the dressing cycles. When the function is active, the control displays a corresponding icon in the general status display.

The following tool types have been added:

- **Grinding wheel,GRIND**
- **Dressing tool,DRESS**

You define the tools in the form view of the tool management. The control displays the tool-specific data on the **TOOL** tab of the additional status display.

8.1.3 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 92

8.1.4 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°.

8.1.5 KinematicsComp (option 52)

Option 52 does not require an export permit anymore.

8.2 New functions

Further information: User's Manual for **Klartext Programming** - 892903-18

Further information: User's Manual for **Setup, Testing and Running NC Programs** - 1261174-11

Further information: User's Manual for **Programming of Machining Cycles** - 892905-18

8.2.1 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.
- Cycle **1000 DEFINE RECIP. STROKE** (ISO: **G1000**, option 156)
This cycle allows you to define a reciprocating stroke for the grinding operation that is superimposed onto the path contour. You define whether the reciprocating stroke starts at the definition of the cycle or after being called later.
- Cycle **1001 START RECIP. STROKE** (ISO: **G1001**, option 156)
This cycle allows you to start a reciprocating stroke that has been defined with Cycle **1000 (G1000)** or stopped with Cycle **1002 (G1002)**.
- Cycle **1002 STOP RECIP. STROKE** (ISO: **G1002**, option 156)
This cycle allows you to stop the active reciprocating stroke. You select whether the control saves or deletes the definition from Cycle **1000 (G1000)**.
- Cycle **1010 DRESSING DIAMETER** (ISO: **G1010**, option 156)
This cycle allows you to dress the diameter of a cylindrical grinding tool. You define the dressing operation and the number of cycle calls after which dressing is performed. You can use this cycle only in dressing mode (**FUNCTION MODE DRESS**).
- Cycle **1015 PROFILE DRESSING** (ISO: **G1015**, option 156)
This cycle allows you to dress a random-shape, rotationally symmetric profile of a grinding tool. You define the profile in a separate NC program. The cycle provides the functions of Cycle **1010 (G1010)** and the option to define an angle of infeed for the dressing operation. You can use this cycle only in dressing mode (**FUNCTION MODE DRESS**).
- Cycle **1030 ACTIVATE WHEEL EDGE** (ISO: **G1030**, option 156)
This cycle allows you to select and activate an edge of the tool for dressing or grinding. You need to define an edge before the dressing operation.

- Cycle **1032 GRINDING WHL LENGTH COMPENSATION** (ISO: **G1032**, option 156)
This cycle allows you to define the overall length of a grinding tool. Depending on whether an initial dressing operation has been performed, you change the compensation data or the basic data of the tool. The compensation data is effective only during grinding, whereas the basic data is also effective during dressing.
- Cycle **1033 GRINDING WHL RADIUS COMPENSATION** (ISO: **G1033**, option 156)
This cycle allows you to define the radius of a grinding tool. Depending on whether an initial dressing operation has been performed, you change the compensation data or the basic data of the tool.
- 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.

8.2.2 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.

8.2.3 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.

8.2.4 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.

8.2.5 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.

8.2.6 Test run

- 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.

8.2.7 Batch Process Manager (option 154)

In the operating modes **Program run, single block** and **Program run, full sequence**, the **COLLISION CHECKING** soft key has been added to the **Batch Process Manager**. In conjunction with option 40, the control simulates all NC programs of a pallet table and checks them for collisions. If the simulation has been completed without a collision, then the control displays a new icon in the **Pgm** column.

8.2.8 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 POSITON CAPTURE** key allows you to load the current position values of an axis into the datum table.

8.2.9 Global Program Settings (option 44)

- The **GLOBAL SETTINGS INACTIVE** soft key that allows you to deactivate all defined settings has been added to the form of the **Global Program Settings** function. After you have pressed this soft key, it changes to **ACTIVATE GLOBAL SETTINGS**. When you press the soft key again, the control activates all of the most recently defined settings.
- While a manual touch probe cycle is being run, the control temporarily deactivates the **Global Program Settings** function.

8.2.10 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 whether the active maximum values of handwheel superimpositioning are defined by **M118** or the **Global Program Settings** function (option 44).

8.2.11 Moving the machine axes

The HR 520 and HR 550 FS handwheels display the offset of handwheel superimpositioning from **M118** or from the **Global Program Settings** function (option 44). The handwheels also show the active setting of the 3-D ROT menu (option 8).

8.2.12 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.

8.2.13 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.

8.2.14 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.
For turning tools (option 50), compensation values from the table *.tco are effective as an alternative to **FUNCTION TURNDATA CORR-TCS**. Compensation values from the table *.wco are effective in the working plane coordinate system as an alternative to **FUNCTION TURNDATA CORR-WPL**.
- 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.

8.2.15 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.

8.2.16 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.

8.2.17 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 **Klartext Programming** - 892903-18

Further information: User's Manual for **Setup, Testing and Running NC Programs** - 1261174-11

Further information: User's Manual for **Programming of Machining Cycles** - 892905-18

8.3.1 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**), 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**) 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).

8.3.2 Touch probe cycles

- Based on the 3-D error (i.e., the deviation along the normal vector), Cycle **444 PROBING IN 3-D** (ISO: **G444**) is able to differentiate between scrap and rework.
- The log of Cycle **444 PROBING IN 3-D** (ISO: **G444**) contains the measured 3-D error.
- 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 semi-automatic mode, then you can use a handwheel for prepositioning and position to a clearance height at the end of the cycle.

8.3.3 Program functions

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

8.3.4 Adaptive Feed Control (AFC) (option 45)

When running the **AFC CUT BEGIN** function in turning mode (option 50), the control checks for whether the programmed rotational speed has been reached. If deviations occur, the control displays an error message and starts the machining operation without monitoring the cutting force.

8.3.5 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.

8.3.6 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.

8.3.7 Test run

- 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

8.3.8 File management

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

8.3.9 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**.

8.3.10 Status displays

- The icon displayed for **FUNCTION MODE TURN** (option 50) in the additional status display has been modified.
- On 19" and 24" screens, 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.

8.3.11 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.

8.3.12 Manual touch probe cycles

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.

8.3.13 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 $\pm 999.9999999^\circ$ to $\pm 99\,999.99999^\circ$.

8.3.14 Tools

- The form view of the tool management shows only those input fields that are needed for the selected tool type.
- The **CUTLENGTH** column of the turning tool table (option 50) contains a default value of 0.

8.3.15 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.

8.3.16 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**).
- When user administration is active, the **Global Program Settings** (option 44) function requires the NC.OPModeMDI 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 34059x-11

9.1 New software options

9.1.1 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** - 892903-19

Further information: User's Manual for **Setup, Testing and Running NC Programs** - 1261174-12

Further information: User's Manual for **Programming of Machining Cycles** - 1303406-10

Further information: User's Manual for **Programming of Measuring Cycles for Workpieces and Tools** - 1303409-10

9.2.1 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

9.2.2 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.
- Cycle **1016 DRESSING OF CUP WHEEL** (ISO: **G1016**, option 156)
You use this cycle to dress the front face of a cup wheel. You define the optional angle for the relief cut in the tool table. You can use this cycle only in dressing mode (**FUNCTION MODE DRESS**).

- Cycle **1025 GRINDING CONTOUR** (ISO: **G1025**, option 156)
The control uses this cycle to grind open or closed contours. You define the contour in a subprogram and select it with Cycle **14 CONTOUR** (ISO: **G37**).
- Cycle **882 SIMULTANEOUS ROUGHING FOR TURNING** (ISO: **G882**, option 50, option 158)
This cycle enables the roughing of turning contours using different inclination angles. This allows you to machine undercut contours, for example, with just one tool. In addition, you can increase the service life of the tool by utilizing a large area of the indexable insert.
You define the contour in a subprogram and select it with Cycle **14 CONTOUR** (ISO: **G37**) or the **SEL CONTOUR** function.

9.2.3 Touch probe cycles

Cycle **485 MEASURE LATHE TOOL** (ISO: **G485**, option 50)

This cycle enables you to measure lathe tools with a tool touch probe. This cycle can only be executed in the **FUNCTION MODE MILL** machining mode. Furthermore, you need a tool touch probe with a cuboid probe contact.

9.2.4 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.

9.2.5 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) ID951**: value in the **SPB-INSERT** column of the turning tool table
 - **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

9.2.6 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.

9.2.7 Test run

- 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
- 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.

9.2.8 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.

9.2.9 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 (option 42) clipboard to an NC program as well as to other applications (e.g., **Leafpad**).

9.2.10 Dynamic Collision Monitoring (DCM) (option 40)

The Fixture Monitoring function (option 40) allows you to integrate fixtures (e.g., a vise) in your NC program. You can use the additional tool **KinematicsDesign** to create the fixtures on the control as CFG files or you can integrate STL files from the CAD system. The control displays the fixtures in the simulation and monitors them for collisions.

9.2.11 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.

9.2.12 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 correctly in the simulation (e.g., neck of end mills or side milling cutters).
- You define an angular offset for recessing tools in the **SPB-INSERT** column of the turning tool table (option 50).
- The **Cup wheel, GRIND_T** tool type has been added for grinding tools.
- You can use M3D and STL files (e.g., from the CAD system) as tool carrier files.

9.2.13 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.

9.2.14 Accessories

You can refit the TE 360 keyboard unit with a HEIDENHAIN 3-D mouse.

9.2.15 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.

9.2.16 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.

9.2.17 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.

9.2.18 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** - 892903-19

Further information: User's Manual for **Setup, Testing and Running NC Programs** - 1261174-12

Further information: User's Manual for **Programming of Machining Cycles** - 1303406-10

Further information: User's Manual for **Programming of Measuring Cycles for Workpieces and Tools** - 1303409-10

9.3.1 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.

9.3.2 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**) 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**)
- Cycle 23 **FLOOR FINISHING** (ISO: **G123**)
- Cycle 24 **SIDE FINISHING** (ISO: **G124**)
- Cycle 233 **FACE MILLING** (ISO: **G233**)
- 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**) 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**) 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) has been enhanced:
 - 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 **272 OCM ROUGHING** (ISO: **G272**, option 167) has been enhanced:
 - 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.
- If a value of 0 is defined in parameter **Q575 INFEEED STRATEGY**, the control machines the individual pockets in a specific sequence. When a pocket has been completed, the control next machines the pocket or boundary with the shortest traverse path from the starting point.
- Cycle **273 OCM FINISHING FLOOR** (ISO: **G273**, option 167) has been enhanced. 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
- You can call the OCM cycles (option 167) with the **CYCL CALL PAT** function (ISO: **G79 PAT**).
- Cycle **1010 DRESSING DIAMETER** (ISO: **G1010**, option 156) uses the value from parameter **Q1018 DRESSING FEED RATE** for the infeed movement.
- In parameter **QS1000 PROFILE PROGRAM** of Cycle **1015 PROFILE DRESSING** (ISO: **G1015**, option 156), you can use the **SELECT FILE** soft key to select the NC program for the profile of the grinding tool.

9.3.3 Touch probe cycles

- Cycles **480 CALIBRATE TT** (ISO: **G480**) and **484 CALIBRATE IR TT** (ISO: **G484**) can be used to calibrate a tool touch probe with cuboid probe contacts.
- For rotating tools, Cycle **483 MEASURE TOOL** (ISO: **G483**) 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**) 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).

9.3.4 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** for look-ahead calculation of a radius-compensated contour is no longer reset by milling cycles

9.3.5 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

9.3.6 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.

9.3.7 Test run

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

9.3.8 CAD Viewer

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

9.3.9 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.

9.3.10 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.

9.3.11 Pallets

In pallet tables, the control can also open NC programs containing blank spaces.

9.3.12 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.

9.3.13 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.

9.3.14 Manual touch probe cycles

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).

9.3.15 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

9.3.16 Tools

In the form view of tool management, the input parameter **TL** is available for grinding and dressing tools. If the check box is enabled, the control locks the tool.

9.3.17 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.

9.3.18 MOD functions

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

9.3.19 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.

9.3.20 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 **AFC** (option 45) functions.
- 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.

9.3.21 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.

10

Software 34059x-16

10.1 New software options

Further information: User's Manual for **Setup, Testing and Running NC Programs** - 1261174-13

10.1.1 CAD Model Optimizer (option 152)

This option allows you to use the **3D mesh** function in the **CAD-Viewer**.

With the **3D mesh** function, you generate STL files from 3D models. This allows you to repair defective files of fixtures and tool holders, for example, or to position STL files generated from the simulation for another machining operation.

10.1.2 Camera-based inspection of the setup situation (VSC)

Visual Setup Control (software option 136) is no longer available.

10.1.3 Production data acquisition

State Reporting Interface (software option 137) is no longer available.

10.2 New functions

Further information: User's Manual for **Klartext Programming** - 892903-1A

Further information: User's Manual for **Setup, Testing and Running NC Programs** - 1261174-13

Further information: User's Manual for **Programming of Machining Cycles** - 1303406-11

Further information: User's Manual for **Programming of Measuring Cycles for Workpieces and Tools** - 1303409-11

10.2.1 NC fundamentals

Within circular paths **C**, **CR** and **CT**, the **LIN_** syntax element is now available in order to superimpose a linear motion over the circular motion of an axis. This allows you to program a helix in a simple way.

In ISO programming, you can define a third axis using the free syntax input in conjunction with the **G02**, **G03** and **G05** functions.

10.2.2 Machining cycles

- Cycle **1017 DRESSING WITH DRESSING ROLL** (ISO: **G1017**, option 156)
This cycle allows you to dress the diameter of a grinding wheel with a dressing roll. Depending on the strategy, the control executes movements based on the wheel geometry. The control provides the dressing strategies reciprocation, oscillating or fine oscillating. You can use this cycle only in dressing mode (**FUNCTION MODE DRESS**).
- Cycle **1018 RECESSING WITH DRESSING ROLL** (ISO: **G1018**, option 156)
This cycle allows you to dress the diameter of a grinding wheel by recessing with a dressing roll. Depending on the strategy, the control executes one or more recessing movements. You can use this cycle only in dressing mode (**FUNCTION MODE DRESS**).
- Cycle **1021 CYLINDER, SLOW-STROKE GRINDING** (ISO: **G1021**, option 156)
This cycle allows you to grind circular pockets or circular studs. The height of the cylinder may exceed the width of the grinding wheel. Through a reciprocating stroke, the control can machine the complete height of the cylinder. The control performs several circular paths during one reciprocating stroke. This process corresponds to grinding with slow stroke.
- Cycle **1022 CYLINDER, FAST-STROKE GRINDING** (ISO: **G1022**, option 156)
This cycle allows you to grind circular pockets or circular studs. The control executes circular and helical paths in order to completely machine the cylindrical surface. To attain the required accuracy and surface definition, you can superimpose a reciprocating stroke onto the movements. This process corresponds to grinding with fast stroke.

10.2.3 Touch probe cycles

- Cycle **1400 POSITION PROBING** (ISO: **G1400**)
This cycle allows you to probe a single position. You can transfer the acquired values to the active line of the preset table.
- Cycle **1401 CIRCLE PROBING** (ISO: **G1401**)
This cycle allows you to determine the center point of a hole or a stud. You can transfer the acquired values to the active line of the preset table.
- Cycle **1402 SPHERE PROBING** (ISO: **G1402**)
This cycle allows you to determine the center point of a sphere. You can transfer the acquired values to the active line of the preset table.
- Cycle **1412 INCLINED EDGE PROBING** (ISO: **G1412**)
This cycle allows you to ascertain a workpiece misalignment by probing two points on an inclined edge.
- Cycle **1493 EXTRUSION PROBING** (ISO: **G1493**)
This cycle allows you to define an extrusion. With an active extrusion, the control repeats the touch points along a direction over a defined length.

10.2.4 Program functions

- With the **TRANS ROTATION** function, you can rotate contours or positions around a rotation angle. The **TRANS DATUM RESET** function allows you to reset the rotation. The NC functions serve as an alternative to Cycle **10 ROTATION**.
- Use the **TRANS MIRROR** function to mirror contours or positions about one or more axes. The **TRANS MIRROR RESET** function allows you to reset the mirroring. The NC functions serve as an alternative to Cycle **8 MIRRORING**.
- With the **TRANS SCALE** function, you can scale contours or positions and thus enlarge or reduce these uniformly. This enables you to program shrinkage and oversize allowances, for example. Use the **TRANS SCALE RESET** function to reset the scaling. The NC functions serve as an alternative to Cycle **11 SCALING FACTOR**.

10.2.5 Programming Q parameters

- Using the **SYNTAX** soft key, you can enclose path information in double quotation marks in order to use any special characters as part of the path, e.g. */*. The control provides the **SYNTAX** soft key with the following NC functions:
 - **FN 16: F-PRINT** (DIN/ISO: **D16**)
 - **FN 26: TABOPEN** (DIN/ISO: **D26**)
- The **FN 18: SYSREAD** (DIN/ISO: **D18**) functions have been enhanced:
 - **FN 18: SYSREAD (D18) ID10**: Read program information
 - **NR8**: Unit of measure of the calling NC program
 - **NR9**: Number of the miscellaneous function
The function is available only within M function macros.
 - **FN 18: SYSREAD (D18) ID210**: Read active coordinate transformations
 - **NR11**: Coordinate system for manual movements
 - **FN 18: SYSREAD (D18) ID295**: Read data of the machine kinematics
 - **NR5**: Type of use of an axis within the kinematics
 - **FN 18: SYSREAD (D18) ID310**: Read geometrical behavior
 - **NR126**: State of the miscellaneous function **M126**

10.2.6 Calling an external NC program

Using the **SYNTAX** soft key, you can enclose path information in double quotation marks in order to use any special characters as part of the path, e.g. */*. The control provides the **SYNTAX** soft key with the following NC functions:

- Cycle **12 PGM CALL** (DIN/ISO: **G39**)
- **CALL PGM** (DIN/ISO: **%**)

10.2.7 Programming aids

The control includes the example tables **WMAT.tab**, **TMAT.tab** and **EXAMPLE.cutd** for automatic cutting data calculation.

10.2.8 File management

Within the file manager, the control hides as well as files with a period at the start of the name. If necessary, you can display the files with the **SHOW FILES** soft key.

10.2.9 CAD Viewer

In the **CAD-Viewer**, you can choose the machining planes **YZ** and **ZX** for milling. You use a selection menu to choose the machining plane.

10.2.10 Program run

The control can execute NC programs with the NC function **SECTION MONITORING**. This NC function may be included with NC programs of the TNC7, but has no function on the TNC 640.

10.2.11 Pallets

You can define a pallet counter on the control. This allows you to define a variable number of parts produced, for example during pallet machining with automatic workpiece change. For this purpose, the columns **TARGET** and **COUNT** were added to the pallet table.

10.2.12 Status displays

The **TRANS** tab of the additional status display contains the active shift in the working plane coordinate system **WPL-CS**. If the shift comes from a compensation table (***.wco**), the control shows the path of the compensation table as well as the number and, if applicable, the comment of the active line.

10.2.13 Tables

The input option TS 760 has been added to the **TYPE** column of the touch probe table.

10.2.14 Tools

- The control makes it possible to define FreeTurn tools and to use them, e.g., for inclined or simultaneous turning operations.
- The **POS. DISP.** soft key enables you to switch the tool table view. The control displays the tool table in combination with the position display or in full-screen mode.

10.2.15 Accessories

The control supports the TS 760 workpiece touch probe.

10.2.16 Machine-specific user parameters

In the machine parameter **speedPosCompType** (no. 403129) the machine manufacturer defines the behavior of speed-controlled FS NC axes when the guard door is open. This allows you, for example, to switch on the workpiece spindle and to touch off the workpiece when the guard door is open.

10.3 Modified functions

Further information: User's Manual for **Klartext Programming** - 892903-1A

Further information: User's Manual for **Setup, Testing and Running NC Programs** - 1261174-13

Further information: User's Manual for **Programming of Machining Cycles** - 1303406-11

Further information: User's Manual for **Programming of Measuring Cycles for Workpieces and Tools** - 1303409-11

10.3.1 NC fundamentals

- In order for the control to show the workpiece blank in the simulation, the workpiece blank must have minimum dimensions. The minimum dimensions are 0.1 mm or 0.004 inches in all axes as well as in the radius.
- The pop-up window for tool selection always shows the content of the **NAME** column, even if you are calling the tool with the tool number.

10.3.2 Machining cycles

- Cycle **225 ENGRAVING** (ISO: **G225**) has been enhanced:
 - The parameter **Q202 MAX. PLUNGING DEPTH** allows you to define the maximum plunging depth.
 - The parameter **Q367 TEXT POSITION** now includes the input options **7, 8** and **9**. With these values, you can set the reference of the engraving text onto the horizontal centerline.
 - The approach behavior was changed. If the tool is below the **2ND SET-UP CLEARANCE**, the control first positions to the 2nd set-up clearance **Q204** and then to the starting position in the working plane.
- The parameter **Q357 CLEARANCE TO SIDE** has been added to Cycle **202 BORING** (ISO: **G202**). This parameter allows you to define how far the control retracts the tool at the bottom of the hole in the working plane. This parameter is only effective if the parameter **Q214 DISENGAGING DIRECTN** has been defined.
- The parameter **Q373 FEED AFTER REMOVAL** has been added to Cycle **205 UNIVERSAL PECKING** (ISO: **G205**). This parameter is used to define the feed rate for returning to the advanced stop distance after chip removal.
- The parameter **Q370 TOOL PATH OVERLAP** has been added to Cycle **208 BORE MILLING** (ISO: **G208**). Use this parameter to define the lateral infeed.
- Cycle **240 CENTERING** (ISO: **G240**) has been enhanced in order to consider pre-drilled diameters.

The following parameters have been added:

- **Q342 ROUGHING DIAMETER**
- **Q253 F PRE-POSITIONING**: If parameter **Q342** is defined, feed rate for approaching the deepened starting point
- If, in Cycle **233 FACE MILLING** (ISO: **G233**), the parameter **Q389** has been defined with the value 2 or 3 and a lateral limit has been defined in addition, then the control approaches the contour or departs from it on an arc with **Q207 FEED RATE MILLING**.
- In Cycle **224 DATAMATRIX CODE PATTERN** (ISO: **G224**), you can output the following system data as variables:
 - Current date
 - Current time
 - Current calendar week number
 - Name and path of an NC program
 - Current count
- The parameters **Q429 COOLANT ON** and **Q430 COOLANT OFF** in Cycle **241 SINGLE-LIP D.H.DRLNG** (ISO: **G241**) have been enhanced. You can define a path for a user macro.

- If a measurement in Cycle **238 MEASURE MACHINE STATUS** (ISO: **G238**, option 155) was not performed correctly, for example at a feed rate override of 0 %, you can repeat the cycle.
- Certain cycles permit you to enter tolerances. In the following cycles, you can define dimensions, tolerances according to DIN EN ISO 286-2 or general tolerances according to DIN ISO 2768-1:
 - Cycle **208 BORE MILLING** (ISO: G208)
 - Cycle **1271 OCM RECTANGLE** (ISO: G1271, option 167)
 - Cycle **1272 OCM CIRCLE** (ISO: G1272, option 167)
 - Cycle **1273 OCM SLOT / RIDGE** (ISO: G1273, option 167)
 - Cycle **1278 OCM POLYGON** (ISO: G1278, option 167)
- An input value of 2 has been added to parameter **Q575 INFEEED STRATEGY** in Cycle **272 OCM ROUGHING** (ISO: **G272**, option 167). With this input option, the control calculates the machining sequence so that the tooth length of the tool is used to its full extent.
- In turning mode with active rotation of the coordinate system (Cycle **800**, option 50), Cycles **286 GEAR HOBGING** (ISO: **G286**, option 157) and **287 GEAR SKIVING** (ISO: **G287**, option 157) automatically calculate a correct retraction direction within a liftoff.
- Cycle **287 GEAR SKIVING** (ISO: **G287**, option 157) has been enhanced:
 - The parameter **Q466 OVERRUN PATH** allows you to define the length of the path at the end point of the gear wheel.
 - An input option for a technology table has been added to the parameter **Q240 NUMBER OF CUTS**. This technology table allows you to define the feed rate, lateral infeed and lateral offset for each individual step.
- Cycle **1010 DRESSING DIAMETER** (ISO: **G1010**, option 156) supports the "dressing roll" tool type.
- You can use Cycle **292 CONTOUR.TURNG.INTRP.** (ISO: **G292**, option 96) with polar kinematics. As a requirement, the workpiece must be clamped in the center of the rotary table and there must be no active coupling.
- Cycle **800 ADJUST XZ SYSTEM** (ISO: **G800**, option 50) has been enhanced:
 - The parameter **Q599 RETRACT** allows you to define a retraction of the tool before positioning movements in the cycle.
 - The cycle considers the miscellaneous function **M138**, Take rotary axes into account during machining operations.
 - The cycle takes into account the angle of a cranked recessing tool.

- The following cycles support machining with a FreeTurn tool:
 - Cycle **811 SHOULDER, LONGITDNL.** (ISO: **G811**, option 50)
 - Cycle **812 SHOULDER, LONG. EXT.** (ISO: **G812**, option 50)
 - Cycle **813 TURN PLUNGE CONTOUR LONGITUDINAL** (ISO: **G813**, option 50)
 - Cycle **814 TURN PLUNGE LONGITUDINAL EXT.** (ISO: **G814**, option 50)
 - Cycle **810 TURN CONTOUR LONG.** (ISO: **G810**, option 50)
 - Cycle **815 CONTOUR-PAR. TURNING** (ISO: **G815**, option 50)
 - Cycle **821 SHOULDER, FACE** (ISO: **G821**, option 50)
 - Cycle **822 SHOULDER, FACE. EXT.** (ISO: **G822**, option 50)
 - Cycle **823 TURN TRANSVERSE PLUNGE** (ISO: **G823**, option 50)
 - Cycle **824 TURN PLUNGE TRANSVERSE EXT.** (ISO: **G824**, option 50)
 - Cycle **820 TURN CONTOUR TRANSV.** (ISO: **G820**, option 50)
 - Cycle **882 SIMULTANEOUS ROUGHING FOR TURNING** (ISO: **G882**, option 158)
 - Cycle **883 TURNING SIMULTANEOUS FINISHING** (ISO: **G882**, option 158)
- The Cycles **860 to 862** and **870 to 872** output an error message during active multiple plunging if diagonal retraction has been programmed (**Q462=1**). Only linear retraction is possible with multiple plunging.

10.3.3 Touch probe cycles

- The Cycles **1420 PROBING IN PLANE** (ISO: **G1420**), **1410 PROBING ON EDGE** (ISO: **G1410**), **1411 PROBING TWO CIRCLES** (ISO: **G1411**) have been enhanced:
 - In these cycles, you can define tolerances according to DIN EN ISO 286-2 or general tolerances according to DIN ISO 2768-1.
 - If you defined the value 2 in parameter **Q1125 CLEAR. HEIGHT MODE**, the control positions the touch probe at rapid traverse **FMAX** from the touch probe table to the set-up clearance.
- The unit of measurement of the main program can be seen in the header of the log file of the Probing Cycles **14xx** and **42x**
- The parameter **Q523 TT-POSITION** has been added to Cycle **484 CALIBRATE IR TT** (ISO: **G484**). This parameter allows you to define the position of the tool touch probe and, if desired, to transfer the position to the machine parameter **centerPos** after calibration.
- If a basic rotation is active in the workpiece preset, the control shows an error message during execution of Cycles **451 MEASURE KINEMATICS** (ISO: **G451**, option 48), **452 PRESET COMPENSATION** (ISO: **G452**, option 48), **453 KINEMATICS GRID** (ISO: **G453**, option 48, option 52). The control resets the basic rotation to 0 during continuation of the program.

10.3.4 Program functions

- Within **FUNCTION S-PULSE**, you can define a lower and an upper speed limit for the pulsing speed by means of the **FROM-SPEED** and **TO-SPEED** syntax elements.
- In the **OPEN FILE** function, the control opens files automatically with the additional tool that was used last for this file type.
- Within the **CONTOUR DEF** function, you can exclude certain areas **V** (void) from machining. These areas may be contours in cast parts or machining operations from previous steps, for example.

10.3.5 Programming Q parameters

While program run is interrupted or has been canceled, you can edit Q and QS parameters with numbers 0 to 99, 200 to 1199 and 1400 to 1999 in the **Q parameter list** window.

10.3.6 Programming aids

The result fields and the diameter field of the cutting data calculator are freely editable.

10.3.7 File management

If you exit an NC program by pressing the **END** key, the control opens the file manager. The cursor is on the NC program that was just closed. If you press the **END** key again, the control opens the original NC program again with the cursor on the last selected line. With large files this behavior can cause a delay.

10.3.8 CAD Viewer

CAD Import (option 42) enables you to use a closed contour as workpiece blank for turning (option 50).

10.3.9 Manual touch probe cycles

- If you use a manual touch-probe function to automatically probe a stud or hole with an opening angle of 360°, the control returns the workpiece touch probe to the starting position at the end of the probing process.
- With the **PROBING PL** function, the control shows a help graphic prior to aligning the 3-D basic rotation in order to point out the danger of collision during tilting.

10.3.10 Tools

- The input range of the **INIT_D_PNR** column in the grinding tool table was expanded from 99 to 9999.
- For the tool type **Dressing roll, ROLL**, the control shows the parameter **CUTWIDTH** in the form view of the tool management.

10.3.11 Error messages and help system

The **FILTER** soft key in the error window was renamed to **GROUPING**. With this soft key, the control groups warnings and error messages.

10.3.12 HEROS functions

- The appearance of the **Network settings** window was changed. For the network configuration, you need to use the **Network Connections** window.
- The control creates certificates for the OPC UA NC Server (options 56 to 61) with a validity of five years.

10.3.13 Machine-specific user parameters

The input range of the machine parameter **displayPace** (no. 101000) has been expanded. The minimum display step of the axes is 0.000001° or mm.

11

Software 34059x-17

11.1 New software options

Further information: User's Manual for **Programming of Machining Cycles** - 1303406-12

11.1.1 Advanced turning functions

Software option 158 was renamed **Turning v2**.

The software option **Turning v2** includes all functions of **Turning** (software option 50) in addition to Cycle **882 SIMULTANEOUS ROUGHING FOR TURNING** and Cycle **883 TURNING SIMULTANEOUS FINISHING**.

11.2 New functions

Further information: User's Manual for **Klartext Programming** - 892903-1B

Further information: User's Manual for **Setup, Testing and Running NC Programs** - 1261174-14

Further information: User's Manual for **Programming of Measuring Cycles for Workpieces and Tools** - 1303409-12

11.2.1 Touch probe cycles

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

11.2.2 Programming Q parameters

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

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

11.2.3 Tables

You define the shape of the stylus in the **STYLUS** column of the touch probe table. You define an L-shaped stylus with the **L-TYPE** selection.

11.2.4 Tools

- The following tool types have been added:
 - **Face milling cutter, MILL_FACE**
 - **Chamfer mill, MILL_CHAMFER**
- You define a database ID for the tool in the **DB_ID** column of the tool table. In a tool database for all machines, you can identify tools with unique database IDs (e.g., within a workshop). This allows you to coordinate the tools of multiple machines more easily.
- You define a radius at the tip of the tool in the **R_TIP** column of the tool table.
- You define the compensation method for dressing in the **COR_TYPE** input parameter for grinding tools (option 156):
 - **Grinding wheel with compensation, COR_TYPE_GRINDTOOL**
Stock removal on the grinding tool
 - **Dressing tool with wear, COR_TYPE_DRESSTOOL**
Stock removal on dressing tool

11.2.5 MOD functions

A link to the **Certificate and keys** HEROS function has been added to the **External access** MOD function. This function can be used to define settings for secure connections via SSH.

11.2.6 HEROS functions

The **OPC UA NC Server** enables client applications to access the tool data of the control. You can read and write tool data.

The **OPC UA NC Server** does not provide access to the grinding and dressing tool tables (option 156).

11.3 Modified functions

Further information: User's Manual for **Klartext Programming** - 892903-1B

Further information: User's Manual for **Setup, Testing and Running NC Programs** - 1261174-14

Further information: User's Manual for **Programming of Machining Cycles** - 1303406-12

Further information: User's Manual for **Programming of Measuring Cycles for Workpieces and Tools** - 1303409-12

11.3.1 Machining cycles

- Cycle **277 OCM CHAMFERING** (ISO: **G277**, option 167) monitors contour damage on the floor caused by the tool tip. This tool tip results from the radius **R**, the radius at the tool tip **R_TIP**, and the point angle **T-ANGLE**.
- The parameter **Q592 TYPE OF DIMENSION** has been added to Cycle **292 CONTOUR.TURNG.INTRP.** (ISO: **G292**, option 96). This parameter is used to define whether the contour is programmed with radius dimensions or diameter dimensions.
- The following cycles consider the miscellaneous functions **M109** and **M110**:
 - Cycle **22 ROUGH-OUT** (ISO: G122)
 - Cycle **23 FLOOR FINISHING** (ISO: G123)
 - Cycle **24 SIDE FINISHING** (ISO: G124)
 - Cycle **25 CONTOUR TRAIN** (ISO: G125)
 - Cycle **275 TROCHOIDAL SLOT** (ISO: G275)
 - Cycle **276 THREE-D CONT. TRAIN** (ISO: G276)
 - Cycle **274 OCM FINISHING SIDE** (ISO: G274, option 167)
 - Cycle **277 OCM CHAMFERING** (ISO: G277, option 167)
 - Cycle **1025 GRINDING CONTOUR** (ISO: G1025, option 156)

11.3.2 Touch probe cycles

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

11.3.3 Program functions

You can use the **TABDATA** functions for read and write access to the preset table.

11.3.4 Test run

- The simulation takes into account the following columns of the tool table:
 - **R_TIP**
 - **LU**
 - **RN**
- The control takes into account the following NC functions in the **Test Run** operating mode.
 - **FN 27: TABWRITE** (ISO: **D27**)
 - **FUNCTION FILE**
 - **FUNCTION FEED DWELL**

11.3.5 CAD Viewer

CAD-Viewer has been enhanced:

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

11.3.6 Monitoring of configured machine components for overload (option 155)

The machine manufacturer can define up to 20 components to be monitored by the control by means of component monitoring.

11.3.7 Moving the machine axes

If a handwheel is active, the control shows the contouring feed rate in the display during program run. If only the currently selected axis is moving, the control shows the axis feed rate.

11.3.8 Tables

The minimum input value of the **FMAX** column in the touch probe table has been changed from -9999 to +10.

11.3.9 Tools

- The maximum input range of the **LTOL** and **RTOL** columns of the tool table has been increased from 0 - 0.9999 mm to 0.0000 - 5.0000 mm.
- The maximum input range of the **LBREAK** and **RBREAK** columns of the tool table has been increased from 0 - 0.9999 mm to 0.0000 - 9.0000 mm.
- In the form view of the tool management, the **HW** checkbox for grinding tools (option 156) was removed.
- For grinding tools of the type **Cup wheel, GRIND_T** you can edit the **ALPHA** parameter.

11.3.10 Accessories

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

11.3.11 HEROS functions

- The HEROS tool **Diffuse** has been removed.
- In the **Certificate and keys** window you can select a file with additional public SSH keys in the **Externally administered SSH key file** area. This allows you to use SSH keys without needing to transmit them to the control.
- You can export and import existing network configurations in the **Network settings** window.

11.3.12 Machine-specific user parameters

The machine manufacturer uses the machine parameters **allowUnsecureLsv2** (no. 135401) and **allowUnsecureRpc** (no. 135402) to define whether the control disables non-secure LSV2 or RPC connections even if user administration is not active. These machine parameters are included in the data object **CfgDncAllowUnsecur** (135400).

When the control detects a non-secure connection, it displays an informational note.

HEIDENHAIN

DR. JOHANNES HEIDENHAIN GmbH

Dr.-Johannes-Heidenhain-Straße 5

83301 Traunreut, Germany

☎ +49 8669 31-0

☎ +49 8669 32-5061

info@heidenhain.de

Technical support ☎ +49 8669 32-1000

Measuring systems ☎ +49 8669 31-3104

service.ms-support@heidenhain.de

NC support ☎ +49 8669 31-3101

service.nc-support@heidenhain.de

NC programming ☎ +49 8669 31-3103

service.nc-pgm@heidenhain.de

PLC programming ☎ +49 8669 31-3102

service.plc@heidenhain.de

APP programming ☎ +49 8669 31-3106

service.app@heidenhain.de

www.heidenhain.com

