





## Contents

1	First Steps with the TNC 640.....	45
2	Introduction.....	67
3	Programming: Fundamentals, file management.....	87
4	Programming: Programming aids.....	127
5	Programming: Tools.....	151
6	Programming: Programming contours.....	187
7	Programming: Data transfer from DXF files or plain-language contours.....	215
8	Programming: Subprograms and program section repeats.....	233
9	Programming: Q Parameters.....	249
10	Programming: Miscellaneous functions.....	313
11	Programming: Special functions.....	333
12	Programming: Multiple Axis Machining.....	367
13	Programming: Pallet editor.....	405
14	Programming: Turning Operations.....	411
15	Manual operation and setup.....	437
16	Positioning with Manual Data Input.....	499
17	Test run and program run.....	505
18	MOD functions.....	533
19	Tables and overviews.....	557



# Contents





<b>1</b>	<b>First Steps with the TNC 640.....</b>	<b>45</b>
1.1	Overview.....	46
1.2	Machine switch-on.....	46
	Acknowledging the power interruption and moving to the reference points.....	46
1.3	Programming the first part.....	47
	Selecting the correct operating mode.....	47
	The most important TNC keys.....	47
	Creating a new program/file management.....	48
	Defining a workpiece blank.....	49
	Program layout.....	50
	Programming a simple contour.....	51
	Creating a cycle program.....	54
1.4	Graphically testing the first part.....	56
	Selecting the correct operating mode.....	56
	Selecting the tool table for the test run.....	56
	Choosing the program you want to test.....	57
	Selecting the screen layout and the view.....	57
	Starting the test run.....	58
1.5	Setting up tools.....	59
	Selecting the correct operating mode.....	59
	Preparing and measuring tools.....	59
	The tool table TOOL.T.....	60
	The pocket table TOOL_PTCH.....	61
1.6	Workpiece setup.....	62
	Selecting the correct operating mode.....	62
	Clamping the workpiece.....	62
	Workpiece alignment with 3-D touch probe.....	63
	Datum setting with 3-D touch probe.....	64
1.7	Running the first program.....	65
	Selecting the correct operating mode.....	65
	Choosing the program you want to run.....	65
	Start the program.....	65

<b>2</b>	<b>Introduction.....</b>	<b>67</b>
2.1	<b>The TNC 640.....</b>	<b>68</b>
	Programming: HEIDENHAIN conversational and ISO formats.....	68
	Compatibility.....	68
2.2	<b>Visual display unit and operating panel.....</b>	<b>69</b>
	Display screen.....	69
	Setting the screen layout.....	69
	Control Panel.....	70
2.3	<b>Modes of Operation.....</b>	<b>71</b>
	Manual Operation and El. Handwheel.....	71
	Positioning with Manual Data Input.....	71
	Programming.....	71
	Test Run.....	72
	Program Run, Full Sequence and Program Run, Single Block.....	72
2.4	<b>Status displays.....</b>	<b>73</b>
	"General" status display.....	73
	Additional status displays.....	74
2.5	<b>Window Manager.....</b>	<b>81</b>
	Task bar.....	82
2.6	<b>SELinux security software.....</b>	<b>83</b>
2.7	<b>Accessories: HEIDENHAIN 3-D Touch Probes and Electronic Handwheels.....</b>	<b>84</b>
	3-D touch probes.....	84
	HR electronic handwheels.....	85



<b>3</b>	<b>Programming: Fundamentals, file management.....</b>	<b>87</b>
<b>3.1</b>	<b>Fundamentals.....</b>	<b>88</b>
	Position encoders and reference marks.....	88
	Reference system.....	88
	Reference system on milling machines.....	89
	Designation of the axes on milling machines.....	89
	Polar coordinates.....	90
	Absolute and incremental workpiece positions.....	91
	Selecting the datum.....	92
<b>3.2</b>	<b>Opening programs and entering.....</b>	<b>93</b>
	Organization of an NC program in DIN/ISO format.....	93
	Define the blank: G30/G31.....	93
	Opening a new part program.....	94
	Programming tool movements in DIN/ISO.....	95
	Actual position capture.....	96
	Editing a program.....	97
	The TNC search function.....	100
<b>3.3</b>	<b>File manager: Fundamentals.....</b>	<b>102</b>
	Files.....	102
	Displaying externally generated files on the TNC.....	104
	Data Backup.....	104

<b>3.4 Working with the file manager.....</b>	<b>105</b>
Directories.....	105
Paths.....	105
Overview: Functions of the file manager.....	106
Calling the file manager.....	107
Selecting drives, directories and files.....	108
Creating a new directory.....	109
Creating a new file.....	109
Copying a single file.....	109
Copying files into another directory.....	110
Copying a table.....	111
Copying a directory.....	112
Choosing one of the last files selected.....	112
Deleting a file.....	113
Deleting a directory.....	113
Tagging files.....	114
Renaming a file.....	115
Sorting files.....	115
Additional functions.....	116
Additional tools for management of external file types.....	117
Data transfer to/from an external data medium.....	122
The TNC in a network.....	124
USB devices on the TNC.....	125

<b>4</b>	<b>Programming: Programming aids.....</b>	<b>127</b>
4.1	<b>Adding comments.....</b>	<b>128</b>
	Application.....	128
	Entering comments during programming.....	128
	Inserting comments after program entry.....	128
	Entering a comment in a separate block.....	128
	Functions for editing of the comment.....	129
4.2	<b>Display of NC Programs.....</b>	<b>130</b>
	Syntax highlighting.....	130
	Scrollbar.....	130
4.3	<b>Structuring programs.....</b>	<b>131</b>
	Definition and applications.....	131
	Displaying the program structure window / Changing the active window.....	131
	Inserting a structuring block in the (left) program window.....	131
	Selecting blocks in the program structure window.....	131
4.4	<b>Calculator.....</b>	<b>132</b>
	Operation.....	132
4.5	<b>Programming graphics.....</b>	<b>134</b>
	Generating / not generating graphics during programming.....	134
	Generating a graphic for an existing program.....	134
	Block number display ON/OFF.....	135
	Erasing the graphic.....	135
	Showing grid lines.....	135
	Magnification or reduction of details.....	136

<b>4.6 Error messages.....</b>	<b>137</b>
Display of errors.....	137
Open the error window.....	137
Closing the error window.....	137
Detailed error messages.....	138
INTERNAL INFO soft key.....	138
Clearing errors.....	139
Error log.....	139
Keystroke log.....	140
Informational texts.....	141
Saving service files.....	141
Calling the TNCguide help system.....	142
<b>4.7 TNCguide context-sensitive help system.....</b>	<b>143</b>
Application.....	143
Working with the TNCguide.....	144
Downloading current help files.....	148



<b>5</b>	<b>Programming: Tools.....</b>	<b>151</b>
5.1	Entering tool-related data.....	152
	Feed rate F.....	152
	Spindle speed S.....	153
5.2	Tool data.....	154
	Requirements for tool compensation.....	154
	Tool number, tool name.....	154
	Tool length L.....	154
	Tool radius R.....	154
	Delta values for lengths and radii.....	155
	Entering tool data into the program.....	155
	Enter tool data into the table.....	156
	Importing tool tables.....	164
	Pocket table for tool changer.....	165
	Call tool data.....	168
	Tool change.....	170
	Tool usage test.....	173
	Tool management (software option).....	175
5.3	Tool compensation.....	182
	Introduction.....	182
	Tool length compensation.....	182
	Tool radius compensation.....	183

<b>6</b>	<b>Programming: Programming contours.....</b>	<b>187</b>
<b>6.1</b>	<b>Tool movements.....</b>	<b>188</b>
	Path functions.....	188
	Miscellaneous functions M.....	188
	Subprograms and program section repeats.....	188
	Programming with Q parameters.....	188
<b>6.2</b>	<b>Fundamentals of Path Functions.....</b>	<b>189</b>
	Programming tool movements for workpiece machining.....	189
<b>6.3</b>	<b>Approaching and departing a contour.....</b>	<b>192</b>
	Starting point and end point.....	192
	Tangential approach and departure.....	194
<b>6.4</b>	<b>Path contours - Cartesian coordinates.....</b>	<b>196</b>
	Overview of path functions.....	196
	Programming path functions.....	196
	Straight line in rapid traverse G00 Straight line with feed rate G01 F.....	197
	Inserting a chamfer between two straight lines.....	198
	Corner rounding G25.....	199
	Circle center I, J.....	200
	Circular path C around circle center CC.....	201
	Circle G02/G03/G05 with defined radius.....	202
	Circle G06 with tangential connection.....	204
	Example: Linear movements and chamfers with Cartesian coordinates.....	205
	Example: Circular movements with Cartesian coordinates.....	206
	Example: Full circle with Cartesian coordinates.....	207
<b>6.5</b>	<b>Path contours – Polar coordinates.....</b>	<b>208</b>
	Overview.....	208
	Zero point for polar coordinates: pole I, J.....	209
	Straight line in rapid traverse G10 Straight line with feed rate G11 F.....	209
	Circular path G12/G13/G15 around pole I, J.....	210
	Circle G16 with tangential connection.....	210
	Helix.....	211
	Example: Linear movement with polar coordinates.....	213
	Example: Helix.....	214



**7 Programming: Data transfer from DXF files or plain-language contours..... 215**

**7.1 Processing DXF Files (Software Option)..... 216**

- Application..... 216
- Opening a DXF file..... 217
- Working with the DXF converter..... 217
- Basic settings..... 218
- Setting layers..... 220
- Defining the datum..... 221
- Selecting and saving a contour..... 223
- Selecting and saving machining positions..... 227

<b>8</b>	<b>Programming: Subprograms and program section repeats.....</b>	<b>233</b>
<b>8.1</b>	<b>Labeling Subprograms and Program Section Repeats.....</b>	<b>234</b>
	Label.....	234
<b>8.2</b>	<b>Subprograms.....</b>	<b>235</b>
	Operating sequence.....	235
	Programming notes.....	235
	Programming a subprogram.....	235
	Calling a subprogram.....	236
<b>8.3</b>	<b>Program-section repeats.....</b>	<b>237</b>
	Label G98.....	237
	Operating sequence.....	237
	Programming notes.....	237
	Programming a program section repeat.....	237
	Calling a program section repeat.....	238
<b>8.4</b>	<b>Any desired program as subprogram.....</b>	<b>239</b>
	Operating sequence.....	239
	Programming notes.....	239
	Calling any program as a subprogram.....	240
<b>8.5</b>	<b>Nesting.....</b>	<b>241</b>
	Types of nesting.....	241
	Nesting depth.....	241
	Subprogram within a subprogram.....	242
	Repeating program section repeats.....	243
	Repeating a subprogram.....	244
<b>8.6</b>	<b>Programming examples.....</b>	<b>245</b>
	Example: Milling a contour in several infeeds.....	245
	Example: Groups of holes.....	246
	Example: Group of holes with several tools.....	247

<b>9</b>	<b>Programming: Q Parameters</b>	<b>249</b>
<b>9.1</b>	<b>Principle and overview of functions</b>	<b>250</b>
	Programming notes	251
	Calling Q parameter functions	252
<b>9.2</b>	<b>Part families—Q parameters in place of numerical values</b>	<b>253</b>
	Application	253
<b>9.3</b>	<b>Describing contours with mathematical functions</b>	<b>254</b>
	Application	254
	Overview	254
	Programming fundamental operations	255
<b>9.4</b>	<b>Angle functions (trigonometry)</b>	<b>256</b>
	Definitions	256
	Programming trigonometric functions	256
<b>9.5</b>	<b>If-then decisions with Q parameters</b>	<b>257</b>
	Application	257
	Unconditional jumps	257
	Programming if-then decisions	257
<b>9.6</b>	<b>Checking and changing Q parameters</b>	<b>258</b>
	Procedure	258
<b>9.7</b>	<b>Additional functions</b>	<b>260</b>
	Overview	260
	D14: Displaying error messages	261
	D18: Reading system data	265
	D19: Transfer values to PLC	274
	D20: NC and PLC synchronization	274
	D29: Transfer values to the PLC	276
	D37 EXPORT	276

<b>9.8</b>	<b>Accessing tables with SQL commands.....</b>	<b>277</b>
	Introduction.....	277
	A transaction.....	278
	Programming SQL commands.....	280
	Overview of the soft keys.....	280
	SQL BIND.....	281
	SQL SELECT.....	282
	SQL FETCH.....	284
	SQL UPDATE.....	285
	SQL INSERT.....	285
	SQL COMMIT.....	286
	SQL ROLLBACK.....	286
<b>9.9</b>	<b>Entering formulas directly.....</b>	<b>287</b>
	Entering formulas.....	287
	Rules for formulas.....	289
	Programming example.....	290
<b>9.10</b>	<b>String parameters.....</b>	<b>291</b>
	String processing functions.....	291
	Assigning string parameters.....	292
	Chain-linking string parameters.....	292
	Converting a numerical value to a string parameter.....	293
	Copying a substring from a string parameter.....	294
	Converting a string parameter to a numerical value.....	295
	Checking a string parameter.....	296
	Finding the length of a string parameter.....	297
	Comparing alphabetic sequence.....	298
	Reading machine parameters.....	299



**9.11 Preassigned Q parameters..... 302**

Values from the PLC: Q100 to Q107..... 302

Active tool radius: Q108..... 302

Tool axis: Q109..... 302

Spindle status: Q110..... 303

Coolant on/off: Q111..... 303

Overlap factor: Q112..... 303

Unit of measurement for dimensions in the program: Q113..... 303

Tool length: Q114..... 303

Coordinates after probing during program run..... 304

Deviation between actual value and nominal value during automatic tool measurement with the TT  
130..... 304

Tilting the working plane with mathematical angles: rotary axis coordinates calculated by the TNC..... 304

Measurement results from touch probe cycles (see also User's Manual for Cycle Programming)..... 305

**9.12 Programming examples..... 307**

Example: Ellipse..... 307

Example: Concave cylinder machined with spherical cutter..... 309

Example: Convex sphere machined with end mill..... 311

<b>10 Programming: Miscellaneous functions.....</b>	<b>313</b>
<b>10.1 Entering miscellaneous functions M and STOP.....</b>	<b>314</b>
Fundamentals.....	314
<b>10.2 M functions for program run inspection, spindle and coolant.....</b>	<b>315</b>
Overview.....	315
<b>10.3 Miscellaneous functions for coordinate data.....</b>	<b>316</b>
Programming machine-referenced coordinates: M91/M92.....	316
Moving to positions in a non-tilted coordinate system with a tilted working plane: M130.....	318
<b>10.4 Miscellaneous functions for path behavior.....</b>	<b>319</b>
Machining small contour steps: M97.....	319
Machining open contour corners: M98.....	320
Feed rate factor for plunging movements: M103.....	321
Feed rate in millimeters per spindle revolution: M136.....	322
Feed rate for circular arcs: M109/M110/M111.....	323
Calculating the radius-compensated path in advance (LOOK AHEAD): M120.....	324
Superimposing handwheel positioning during program run: M118.....	326
Retraction from the contour in the tool-axis direction: M140.....	328
Suppressing touch probe monitoring: M141.....	329
Deleting basic rotation: M143.....	330
Automatically retract tool from the contour at an NC stop: M148.....	331
Rounding corners: M197.....	332



<b>11 Programming: Special functions.....</b>	<b>333</b>
<b>11.1 Overview of special functions.....</b>	<b>334</b>
Main menu for SPEC FCT special functions.....	334
Program defaults menu.....	335
Functions for contour and point machining menu.....	335
Menu of various DIN/ISO functions.....	336
<b>11.2 Dynamic Collision Monitoring (software option).....</b>	<b>337</b>
Function.....	337
Collision monitoring in the manual operating modes.....	339
Collision monitoring in Automatic operation.....	341
Graphic depiction of the protected space.....	342
<b>11.3 Adaptive Feed Control Software Option (AFC).....</b>	<b>343</b>
Application.....	343
Defining the AFC basic settings.....	345
Recording a teach-in cut.....	348
Activating/deactivating AFC.....	351
Log file.....	352
Tool breakage/tool wear monitoring.....	353
Spindle load monitoring.....	354
<b>11.4 Active Chatter Control (ACC; software option).....</b>	<b>355</b>
Application.....	355
Activating/deactivating ACC.....	355
<b>11.5 Defining DIN/ISO Functions.....</b>	<b>356</b>
Overview.....	356
<b>11.6 Creating Text Files.....</b>	<b>357</b>
Application.....	357
Opening and exiting text files.....	357
Editing texts.....	358
Deleting and re-inserting characters, words and lines.....	358
Editing text blocks.....	359
Finding text sections.....	360

<b>11.7 Freely definable tables.....</b>	<b>361</b>
Fundamentals.....	361
Creating a freely definable table.....	361
Editing the table format.....	362
Switching between table and form view.....	363
D26: TAOPEN: Open a freely definable table.....	364
D27: TAPWRITE: Write to a freely definable table.....	365
D28: TAPREAD: Read from a freely definable table.....	366



<b>12 Programming: Multiple Axis Machining.....</b>	<b>367</b>
<b>12.1 Functions for multiple axis machining.....</b>	<b>368</b>
<b>12.2 The PLANE Function: Tilting the Working Plane (Software Option 1).....</b>	<b>369</b>
Introduction.....	369
Defining the PLANE function.....	371
Position display.....	371
Resetting the PLANE function.....	372
Defining the working plane with the spatial angle: PLANE SPATIAL.....	373
Defining the working plane with the projection angle: PLANE PROJECTED.....	375
Defining the working plane with the Euler angle: PLANE EULER.....	376
Defining the working plane with two vectors: PLANE VECTOR.....	378
Defining the working plane via three points: PLANE POINTS.....	380
Defining the working plane via a single incremental spatial angle: PLANE SPATIAL.....	382
Tilting the working plane through axis angle: PLANE AXIAL (FCL 3 function).....	383
Specifying the positioning behavior of the PLANE function.....	385
<b>12.3 Inclined-tool machining in a tilted machining plane (software option 2).....</b>	<b>390</b>
Function.....	390
Inclined-tool machining via incremental traverse of a rotary axis.....	390
<b>12.4 Miscellaneous functions for rotary axes.....</b>	<b>391</b>
Feed rate in mm/min on rotary axes A, B, C: M116 (software option 1).....	391
Shortest-path traverse of rotary axes: M126.....	392
Reducing display of a rotary axis to a value less than 360°: M94.....	393
Maintaining the position of the tool tip when positioning with tilted axes (TCPM): M128 (software option 2).....	394
Selecting tilting axes: M138.....	397
Compensating the machine's kinematics configuration for ACTUAL/NOMINAL positions at end of block: M144 (software option 2).....	398
<b>12.5 FUNCTION TCPM (software option 2).....</b>	<b>399</b>
Function.....	399
Defining the TCPM FUNCTION.....	399
Mode of action of the programmed feed rate.....	400
Interpretation of the programmed rotary axis coordinates.....	400
Type of interpolation between the starting and end position.....	402
Resetting the TCPM FUNCTION.....	403

<b>12.6 Peripheral Milling: 3-D radius compensation with TCPM and radius compensation (G41/ G42).....</b>	<b>404</b>
Application.....	404



**13 Programming: Pallet editor..... 405**

**13.1 Pallet Management..... 406**

Application..... 406

Select pallet table..... 408

Exiting the pallet file..... 408

Run pallet file..... 408

<b>14 Programming: Turning Operations.....</b>	<b>411</b>
<b>14.1 Turning Operations on Milling Machines (Software Option 50).....</b>	<b>412</b>
Introduction.....	412
<b>14.2 Basis Functions (Software Option 50).....</b>	<b>413</b>
Switching between milling/turning mode of operation.....	413
Graphical display of turning operations.....	415
Program spindle speed.....	416
Feed rate.....	417
Tool call.....	417
Tool compensation in the program.....	418
Tool data.....	419
Tool tip radius compensation TRC.....	424
Recessing and undercutting.....	425
Inclined turning.....	431
<b>14.3 Unbalance Functions.....</b>	<b>433</b>
Unbalance while turning.....	433
Measure Unbalance cycle.....	435



<b>15 Manual operation and setup.....</b>	<b>437</b>
<b>15.1 Switch-on, switch-off.....</b>	<b>438</b>
Switch-on.....	438
Switch-off.....	440
<b>15.2 Moving the machine axes.....</b>	<b>441</b>
Note.....	441
Moving the axis with the machine axis direction buttons.....	441
Incremental jog positioning.....	441
Traverse with electronic handwheels.....	442
<b>15.3 Spindle speed S, feed rate F and miscellaneous function M.....</b>	<b>452</b>
Application.....	452
Entering values.....	452
Adjusting spindle speed and feed rate.....	453
Activating feed-rate limitation.....	453
<b>15.4 Functional safety FS (option).....</b>	<b>454</b>
Miscellaneous.....	454
Explanation of terms.....	455
Checking the axis positions.....	456
Overview of permitted feed rates and speeds.....	457
Activating feed-rate limitation.....	457
Additional status displays.....	458
<b>15.5 Datum setting without a 3-D touch probe.....</b>	<b>459</b>
Note.....	459
Preparation.....	459
Workpiece presetting with axis keys.....	459
Datum management with the preset table.....	460
<b>15.6 Using 3-D touch probes.....</b>	<b>466</b>
Overview.....	466
Functions in touch probe cycles.....	468
Selecting touch probe cycles.....	470
Recording measured values from the touch-probe cycles.....	471
Writing measured values from the touch probe cycles in a datum table.....	472
Writing measured values from the touch probe cycles in the preset table.....	473

<b>15.7</b>	<b>Calibrating a 3-D touch trigger probe.....</b>	<b>474</b>
	Introduction.....	474
	Calibrating the effective length.....	475
	Calibrating the effective radius and compensating center misalignment.....	476
	Displaying calibration values.....	478
<b>15.8</b>	<b>Compensating workpiece misalignment with 3-D touch probe.....</b>	<b>479</b>
	Introduction.....	479
	Identifying basic rotation.....	480
	Saving a basic rotation in the preset table.....	480
	Compensation of workpiece misalignment by rotating the table.....	480
	Displaying a basic rotation.....	481
	Canceling a basic rotation.....	481
<b>15.9</b>	<b>Datum Setting with 3-D Touch Probe.....</b>	<b>482</b>
	Overview.....	482
	Datum setting in any axis.....	482
	Corner as datum.....	483
	Circle center as datum.....	485
	Setting a center line as datum.....	487
	Measuring workpieces with a 3-D touch probe.....	488
	Using touch probe functions with mechanical probes or measuring dials.....	491
<b>15.10</b>	<b>Tilting the working plane (software option 1).....</b>	<b>492</b>
	Application, function.....	492
	Traversing reference points in tilted axes.....	494
	Position display in a tilted system.....	494
	Limitations on working with the tilting function.....	494
	To activate manual tilting:.....	495
	Setting the current tool-axis direction as the active machining direction.....	496
	Setting the datum in a tilted coordinate system.....	497



<b>16 Positioning with Manual Data Input.....</b>	<b>499</b>
<b>16.1 Programming and executing simple machining operations.....</b>	<b>500</b>
Positioning with manual data input (MDI).....	500
Protecting and erasing programs in \$MDI.....	503

<b>17 Test run and program run.....</b>	<b>505</b>
<b>17.1 Graphics.....</b>	<b>506</b>
Application.....	506
Speed of the Setting test runs.....	507
Overview: Display modes.....	508
Plan view.....	509
Projection in three planes.....	509
3-D view.....	510
Magnifying details.....	512
Repeating graphic simulation.....	513
Tool display.....	513
Measurement of machining time.....	514
3-D line graphics.....	515
<b>17.2 Showing the workpiece blank in the working space.....</b>	<b>517</b>
Application.....	517
<b>17.3 Functions for program display.....</b>	<b>518</b>
Overview.....	518
<b>17.4 Test Run.....</b>	<b>519</b>
Application.....	519
<b>17.5 Program run.....</b>	<b>522</b>
Application.....	522
Running a part program.....	523
Interrupt machining.....	524
Moving the machine axes during an interruption.....	525
Resuming program run after an interruption.....	525
Any entry into program (mid-program startup).....	527
Returning to the contour.....	529
<b>17.6 Automatic program start.....</b>	<b>530</b>
Application.....	530
<b>17.7 Optional block skip.....</b>	<b>531</b>
Application.....	531
Inserting the "/" character.....	531
Erasing the "/" character.....	531



**17.8 Optional program-run interruption.....532**

Application..... 532

<b>18 MOD functions.....</b>	<b>533</b>
<b>18.1 MOD function.....</b>	<b>534</b>
Selecting MOD functions.....	534
Changing the settings.....	534
Exiting MOD functions.....	534
Overview of MOD functions.....	535
<b>18.2 Machine settings.....</b>	<b>536</b>
External access.....	536
Select kinematics.....	538
<b>18.3 Position Display Types.....</b>	<b>539</b>
Application.....	539
<b>18.4 Unit of Measurement.....</b>	<b>540</b>
Application.....	540
<b>18.5 Displaying operating times.....</b>	<b>540</b>
Application.....	540
<b>18.6 Software numbers.....</b>	<b>541</b>
Application.....	541
<b>18.7 Entering the code number.....</b>	<b>541</b>
Application.....	541
<b>18.8 Setting up data interfaces.....</b>	<b>542</b>
Serial interfaces on the TNC 640.....	542
Application.....	542
Setting the RS-232 interface.....	542
Setting the BAUD RATE (baudRate).....	542
Setting the protocol (protocol).....	543
Setting data bits (dataBits).....	543
Check parity (parity).....	543
Setting the stop bits (stopBits).....	543
Setting handshaking (flowControl).....	544
File system for file operations (fileSystem).....	544
Settings for data transfer with the TNCserver PC software.....	544
Setting the operating mode of the external device (fileSystem).....	545
Data transfer software.....	546



**18.9 Ethernet interface..... 548**

Introduction..... 548  
Connection options..... 548  
Configuring the TNC..... 548

**18.10 Configure HR 550 FS wireless handwheel..... 554**

Application..... 554  
Assigning the handwheel to a specific handwheel holder..... 554  
Setting the transmission channel..... 555  
Selecting the transmitter power..... 555  
Statistical data..... 556

<b>19 Tables and overviews.....</b>	<b>557</b>
<b>19.1 Machine-specific user parameters.....</b>	<b>558</b>
Application.....	558
<b>19.2 Connector pin layout and connection cables for data interfaces.....</b>	<b>568</b>
RS-232-C/V.24 interface for HEIDENHAIN devices.....	568
Non-HEIDENHAIN devices.....	570
Ethernet interface RJ45 socket.....	570
<b>19.3 Technical Information.....</b>	<b>571</b>
<b>19.4 Overview tables.....</b>	<b>579</b>
Fixed cycles.....	579
Miscellaneous functions.....	580
<b>19.5 Functions of the TNC 640 and the iTNC 530 compared.....</b>	<b>582</b>
Comparison: Specifications.....	582
Comparison: Data interfaces.....	582
Comparison: Accessories.....	583
Comparison: PC software.....	583
Comparison: Machine-specific functions.....	584
Comparison: User functions.....	584
Comparison: Cycles.....	591
Comparison: Miscellaneous functions.....	593
Comparison: Touch probe cycles in the Manual Operation and El. Handwheel modes.....	595
Comparison: Touch probe cycles for automatic workpiece inspection.....	596
Comparison: Differences in programming.....	597
Comparison: Differences in Test Run, functionality.....	601
Comparison: Differences in Test Run, operation.....	601
Comparison: Differences in Manual Operation, functionality.....	601
Comparison: Differences in Manual Operation, operation.....	603
Comparison: Differences in Program Run, operation.....	603
Comparison: Differences in Program Run, traverse movements.....	604
Comparison: Differences in MDI operation.....	608
Comparison: Differences in programming station.....	609
<b>19.6 DIN/ISO Function Overview TNC 640.....</b>	<b>610</b>