

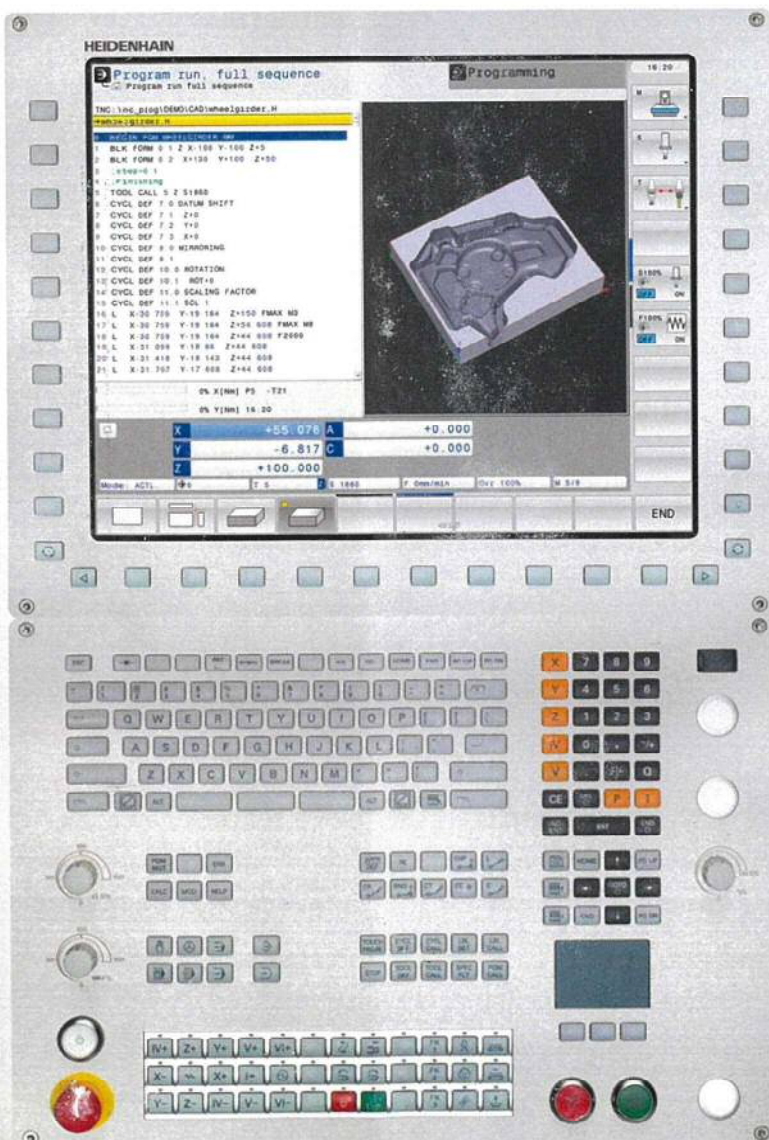
HEIDENHAIN

TNC 640 User's Manual

HEIDENHAIN
Conversational Programming

NC Software
340590-02
340591-02
340594-02

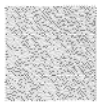
English (en)
5/2013





Contents

1	First Steps with the TNC 640.....	47
2	Introduction.....	69
3	Programming: Fundamentals, file management.....	89
4	Programming: Programming aids.....	131
5	Programming: Tools.....	155
6	Programming: Programming contours.....	191
7	Programming: Data transfer from DXF files or plain-language contours.....	243
8	Programming: Subprograms and program section repeats.....	261
9	Programming: Q Parameters.....	277
10	Programming: Miscellaneous functions.....	347
11	Programming: Special functions.....	367
12	Programming: Multiple Axis Machining.....	409
13	Programming: Pallet editor.....	453
14	Programming: Turning Operations.....	459
15	Manual operation and setup.....	485
16	Positioning with Manual Data Input.....	547
17	Test run and program run.....	553
18	MOD functions.....	581
19	Tables and overviews.....	605



1	First Steps with the TNC 640	47
1.1	Overview	48
1.2	Machine switch-on	48
	Acknowledging the power interruption and moving to the reference points	48
1.3	Programming the first part	49
	Selecting the correct operating mode	49
	The most important TNC keys	49
	Creating a new program/file management	50
	Defining a workpiece blank	51
	Program layout	52
	Programming a simple contour	53
	Creating a cycle program	56
1.4	Graphically testing the first part	58
	Selecting the correct operating mode	58
	Selecting the tool table for the test run	58
	Choosing the program you want to test	59
	Selecting the screen layout and the view	59
	Starting the test run	60
1.5	Setting up tools	61
	Selecting the correct operating mode	61
	Preparing and measuring tools	61
	The tool table TOOL.T	62
	The pocket table TOOL_PTCH	63
1.6	Workpiece setup	64
	Selecting the correct operating mode	64
	Clamping the workpiece	64
	Workpiece alignment with 3-D touch probe	65
	Datum setting with 3-D touch probe	66
1.7	Running the first program	67
	Selecting the correct operating mode	67
	Choosing the program you want to run	67
	Start the program	67

2	Introduction.....	69
2.1	The TNC 640.....	70
	Programming: HEIDENHAIN conversational and ISO formats.....	70
	Compatibility.....	70
2.2	Visual display unit and operating panel.....	71
	Display screen.....	71
	Setting the screen layout.....	71
	Control Panel.....	72
2.3	Modes of Operation.....	73
	Manual Operation and EI. Handwheel.....	73
	Positioning with Manual Data Input.....	73
	Programming.....	73
	Test Run.....	74
	Program Run, Full Sequence and Program Run, Single Block.....	74
2.4	Status displays.....	75
	"General" status display.....	75
	Additional status displays.....	76
2.5	Window Manager.....	83
	Task bar.....	84
2.6	SELinux security software.....	85
2.7	Accessories: HEIDENHAIN 3-D Touch Probes and Electronic Handwheels.....	86
	3-D touch probes.....	86
	HR electronic handwheels.....	87



3	Programming: Fundamentals, file management.....	89
3.1	Fundamentals.....	90
	Position encoders and reference marks.....	90
	Reference system.....	90
	Reference system on milling machines.....	91
	Designation of the axes on milling machines.....	91
	Polar coordinates.....	92
	Absolute and incremental workpiece positions.....	93
	Selecting the datum.....	94
3.2	Opening programs and entering.....	95
	Organization of an NC program in HEIDENHAIN Conversational format.....	95
	Define the blank: BLK FORM.....	95
	Opening a new part program.....	96
	Programming tool movements in conversational.....	97
	Actual position capture.....	99
	Editing a program.....	100
	The TNC search function.....	103
3.3	File manager: Fundamentals.....	105
	Files.....	105
	Displaying externally generated files on the TNC.....	107
	Data Backup.....	107

3.4 Working with the file manager.....	108
Directories.....	108
Paths.....	108
Overview: Functions of the file manager.....	109
Calling the file manager.....	110
Selecting drives, directories and files.....	111
Creating a new directory.....	112
Creating a new file.....	112
Copying a single file.....	112
Copying files into another directory.....	113
Copying a table.....	114
Copying a directory.....	115
Choosing one of the last files selected.....	115
Deleting a file.....	116
Deleting a directory.....	116
Tagging files.....	117
Renaming a file.....	118
Sorting files.....	118
Additional functions.....	119
Additional tools for management of external file types.....	120
Data transfer to/from an external data medium.....	125
The TNC in a network.....	127
USB devices on the TNC.....	128



4	Programming: Programming aids.....	131
4.1	Adding comments.....	132
	Application.....	132
	Entering comments during programming.....	132
	Inserting comments after program entry.....	132
	Entering a comment in a separate block.....	132
	Functions for editing of the comment.....	133
4.2	Display of NC Programs.....	134
	Syntax highlighting.....	134
	Scrollbar.....	134
4.3	Structuring programs.....	135
	Definition and applications.....	135
	Displaying the program structure window / Changing the active window.....	135
	Inserting a structuring block in the (left) program window.....	135
	Selecting blocks in the program structure window.....	135
4.4	Calculator.....	136
	Operation.....	136
4.5	Programming graphics.....	138
	Generating / not generating graphics during programming.....	138
	Generating a graphic for an existing program.....	138
	Block number display ON/OFF.....	139
	Erasing the graphic.....	139
	Showing grid lines.....	139
	Magnification or reduction of details.....	140

4.6 Error messages..... 141

Display of errors.....	141
Open the error window.....	141
Closing the error window.....	141
Detailed error messages.....	142
INTERNAL INFO soft key.....	142
Clearing errors.....	143
Error log.....	143
Keystroke log.....	144
Informational texts.....	145
Saving service files.....	145
Calling the TNCguide help system.....	146

4.7 TNCguide context-sensitive help system..... 147

Application.....	147
Working with the TNCguide.....	148
Downloading current help files.....	152



5	Programming: Tools	155
5.1	Entering tool-related data	156
	Feed rate F	156
	Spindle speed S	157
5.2	Tool data	158
	Requirements for tool compensation	158
	Tool number, tool name	158
	Tool length L	158
	Tool radius R	158
	Delta values for lengths and radii	159
	Entering tool data into the program	159
	Enter tool data into the table	160
	Importing tool tables	169
	Pocket table for tool changer	170
	Call tool data	173
	Tool change	175
	Tool usage test	178
	Tool management (software option)	180
5.3	Tool compensation	187
	Introduction	187
	Tool length compensation	187
	Tool radius compensation	188

6	Programming: Programming contours.....	191
6.1	Tool movements.....	192
	Path functions.....	192
	FK free contour programming.....	192
	Miscellaneous functions M.....	192
	Subprograms and program section repeats.....	193
	Programming with Q parameters.....	193
6.2	Fundamentals of Path Functions.....	194
	Programming tool movements for workpiece machining.....	194
6.3	Approaching and departing a contour.....	198
	Overview: Types of paths for contour approach and departure.....	198
	Important positions for approach and departure.....	199
	Approaching on a straight line with tangential connection: APPR LT.....	201
	Approaching on a straight line perpendicular to the first contour point: APPR LN.....	201
	Approaching on a circular path with tangential connection: APPR CT.....	202
	Approaching on a circular path with tangential connection from a straight line to the contour: APPR LCT.....	203
	Departing in a straight line with tangential connection: DEP LT.....	204
	Departing in a straight line perpendicular to the last contour point: DEP LN.....	204
	Departing on a circular path with tangential connection: DEP CT.....	205
	Departing on a circular arc tangentially connecting the contour and a straight line: DEP LCT.....	205
6.4	Path contours - Cartesian coordinates.....	206
	Overview of path functions.....	206
	Straight line L.....	207
	Inserting a chamfer between two straight lines.....	208
	Corner rounding RND.....	209
	Circle center CC.....	210
	Circular path C around circle center CC.....	211
	Circle CR with defined radius.....	212
	Circle CT with tangential connection.....	214
	Example: Linear movements and chamfers with Cartesian coordinates.....	215
	Example: Circular movements with Cartesian coordinates.....	216
	Example: Full circle with Cartesian coordinates.....	217



6.5 Path contours – Polar coordinates..... 218

Overview..... 218

Zero point for polar coordinates: pole CC.....219

Straight line LP.....219

Circular path CP around pole CC.....220

Circle CTP with tangential connection.....220

Helix.....221

Example: Linear movement with polar coordinates..... 223

Example: Helix.....224

6.6 Path contours – FK free contour programming..... 225

Fundamentals..... 225

FK programming graphics.....227

Initiating the FK dialog..... 229

Pole for FK programming.....229

Free straight line programming.....230

Free circular path programming.....231

Input options.....232

Auxiliary points.....235

Relative data.....236

Example: FK programming 1.....238

Example: FK programming 2.....239

Example: FK programming 3.....240

7	Programming: Data transfer from DXF files or plain-language contours.....	243
7.1	Processing DXF Files (Software Option).....	244
	Application.....	244
	Opening a DXF file.....	245
	Working with the DXF converter.....	245
	Basic settings.....	246
	Setting layers.....	248
	Defining the datum.....	249
	Selecting and saving a contour.....	251
	Selecting and saving machining positions.....	255

8	Programming: Subprograms and program section repeats	261
8.1	Labeling Subprograms and Program Section Repeats	262
	Label	262
8.2	Subprograms	263
	Operating sequence	263
	Programming notes	263
	Programming a subprogram	263
	Calling a subprogram	264
8.3	Program-section repeats	265
	Label LBL	265
	Operating sequence	265
	Programming notes	265
	Programming a program section repeat	265
	Calling a program section repeat	266
8.4	Any desired program as subprogram	267
	Operating sequence	267
	Programming notes	267
	Calling any program as a subprogram	268
8.5	Nesting	269
	Types of nesting	269
	Nesting depth	269
	Subprogram within a subprogram	270
	Repeating program section repeats	271
	Repeating a subprogram	272
8.6	Programming examples	273
	Example: Milling a contour in several infeeds	273
	Example: Groups of holes	274
	Example: Group of holes with several tools	275

9	Programming: Q Parameters.....	277
9.1	Principle and overview of functions.....	278
	Programming notes.....	279
	Calling Q parameter functions.....	280
9.2	Part families—Q parameters in place of numerical values.....	281
	Application.....	281
9.3	Describing contours with mathematical functions.....	282
	Application.....	282
	Overview.....	282
	Programming fundamental operations.....	283
9.4	Angle functions (trigonometry).....	284
	Definitions.....	284
	Programming trigonometric functions.....	284
9.5	Calculation of circles.....	285
	Application.....	285
9.6	If-then decisions with Q parameters.....	286
	Application.....	286
	Unconditional jumps.....	286
	Programming if-then decisions.....	286
	Abbreviations used:.....	287
9.7	Checking and changing Q parameters.....	288
	Procedure.....	288
9.8	Additional functions.....	290
	Overview.....	290
	FN 14: ERROR: Displaying error messages.....	291
	FN 16: F-PRINT: Output of formatted texts and Q parameter values.....	295
	FN 18: SYS-DATUM READ: Reading system data.....	299
	FN 19: PLC: Transfer values to PLC.....	308
	FN 20: WAIT FOR: NC and PLC synchronization.....	308
	FN 29: PLC: Transfer values to the PLC.....	310
	FN 37: EXPORT.....	310



9.9 Accessing tables with SQL commands..... 311

- Introduction.....311
- A transaction..... 312
- Programming SQL commands..... 314
- Overview of the soft keys..... 314
- SQL BIND.....315
- SQL SELECT..... 316
- SQL FETCH..... 318
- SQL UPDATE..... 319
- SQL INSERT..... 319
- SQL COMMIT..... 320
- SQL ROLLBACK..... 320

9.10 Entering formulas directly..... 321

- Entering formulas..... 321
- Rules for formulas..... 323
- Programming example..... 324

9.11 String parameters..... 325

- String processing functions..... 325
- Assigning string parameters..... 326
- Chain-linking string parameters..... 326
- Converting a numerical value to a string parameter..... 327
- Copying a substring from a string parameter..... 328
- Converting a string parameter to a numerical value..... 329
- Checking a string parameter..... 330
- Finding the length of a string parameter..... 331
- Comparing alphabetic sequence..... 332
- Reading machine parameters..... 333

9.12 Preassigned Q parameters..... 336

Values from the PLC: Q100 to Q107..... 336

Active tool radius: Q108..... 336

Tool axis: Q109..... 336

Spindle status: Q110..... 337

Coolant on/off: Q111..... 337

Overlap factor: Q112..... 337

Unit of measurement for dimensions in the program: Q113..... 337

Tool length: Q114..... 337

Coordinates after probing during program run..... 338

Deviation between actual value and nominal value during automatic tool measurement with the TT 130..... 338

Tilting the working plane with mathematical angles: rotary axis coordinates calculated by the TNC..... 338

Measurement results from touch probe cycles (see also User's Manual for Cycle Programming)..... 339

9.13 Programming examples..... 341

Example: Ellipse..... 341

Example: Concave cylinder machined with spherical cutter..... 343

Example: Convex sphere machined with end mill..... 345



10 Programming: Miscellaneous functions.....	347
10.1 Entering miscellaneous functions M and STOP.....	348
Fundamentals.....	348
10.2 M functions for program run inspection, spindle and coolant.....	349
Overview.....	349
10.3 Miscellaneous functions for coordinate data.....	350
Programming machine-referenced coordinates: M91/M92.....	350
Moving to positions in a non-tilted coordinate system with a tilted working plane: M130.....	352
10.4 Miscellaneous functions for path behavior.....	353
Machining small contour steps: M97.....	353
Machining open contour corners: M98.....	354
Feed rate factor for plunging movements: M103.....	355
Feed rate in millimeters per spindle revolution: M136.....	356
Feed rate for circular arcs: M109/M110/M111.....	357
Calculating the radius-compensated path in advance (LOOK AHEAD): M120.....	358
Superimposing handwheel positioning during program run: M118.....	360
Retraction from the contour in the tool-axis direction: M140.....	362
Suppressing touch probe monitoring: M141.....	363
Deleting basic rotation: M143.....	364
Automatically retract tool from the contour at an NC stop: M148.....	365
Rounding corners: M197.....	366

11 Programming: Special functions.....	367
11.1 Overview of special functions.....	368
Main menu for SPEC FCT special functions.....	368
Program defaults menu.....	369
Functions for contour and point machining menu.....	369
Menu of various conversational functions.....	370
11.2 Dynamic Collision Monitoring (software option).....	371
Function.....	371
Collision monitoring in the manual operating modes.....	373
Collision monitoring in Automatic operation.....	375
Graphic depiction of the protected space.....	376
11.3 Adaptive Feed Control Software Option (AFC).....	377
Application.....	377
Defining the AFC basic settings.....	379
Recording a teach-in cut.....	382
Activating/deactivating AFC.....	385
Log file.....	386
Tool breakage/tool wear monitoring.....	387
Spindle load monitoring.....	388
11.4 Active Chatter Control (ACC; software option).....	389
Application.....	389
Activating/deactivating ACC.....	389
11.5 Working with the Parallel Axes U, V and W.....	390
Overview.....	390
FUNCTION PARAXCOMP DISPLAY.....	391
FUNCTION PARAXCOMP MOVE.....	391
FUNCTION PARAXCOMP OFF.....	392
FUNCTION PARAXMODE.....	392
FUNCTION PARAXMODE OFF.....	393
11.6 File functions.....	394
Application.....	394
Defining file functions.....	394



11.7 Definition of a datum shift.....	395
Overview.....	395
TRANS DATUM AXIS.....	395
TRANS DATUM TABLE.....	396
TRANS DATUM RESET.....	397
11.8 Creating Text Files.....	398
Application.....	398
Opening and exiting text files.....	398
Editing texts.....	399
Deleting and re-inserting characters, words and lines.....	399
Editing text blocks.....	400
Finding text sections.....	401
11.9 Freely definable tables.....	402
Fundamentals.....	402
Creating a freely definable table.....	402
Editing the table format.....	403
Switching between table and form view.....	404
FN 26: TAOPEN: Open a freely definable table.....	405
FN 27: TAPWRITE: Write to a freely definable table.....	406
FN 28: TAPREAD: Read from a freely definable table.....	407

12 Programming: Multiple Axis Machining.....	409
12.1 Functions for multiple axis machining.....	410
12.2 The PLANE Function: Tilting the Working Plane (Software Option 1).....	411
Introduction.....	411
Defining the PLANE function.....	413
Position display.....	413
Resetting the PLANE function.....	414
Defining the working plane with the spatial angle: PLANE SPATIAL.....	415
Defining the working plane with the projection angle: PLANE PROJECTED.....	417
Defining the working plane with the Euler angle: PLANE EULER.....	418
Defining the working plane with two vectors: PLANE VECTOR.....	420
Defining the working plane via three points: PLANE POINTS.....	422
Defining the working plane via a single incremental spatial angle: PLANE SPATIAL.....	424
Tilting the working plane through axis angle: PLANE AXIAL (FCL 3 function).....	425
Specifying the positioning behavior of the PLANE function.....	427
12.3 Inclined-tool machining in a tilted machining plane (software option 2).....	432
Function.....	432
Inclined-tool machining via incremental traverse of a rotary axis.....	432
Inclined-tool machining via normal vectors.....	433
12.4 Miscellaneous functions for rotary axes.....	434
Feed rate in mm/min on rotary axes A, B, C: M116 (software option 1).....	434
Shortest-path traverse of rotary axes: M126.....	435
Reducing display of a rotary axis to a value less than 360°: M94.....	436
Maintaining the position of the tool tip when positioning with tilted axes (TCPM): M128 (software option 2).....	437
Selecting tilting axes: M138.....	440
Compensating the machine's kinematics configuration for ACTUAL/NOMINAL positions at end of block: M144 (software option 2).....	441
12.5 FUNCTION TCPM (software option 2).....	442
Function.....	442
Defining the TCPM FUNCTION.....	442
Mode of action of the programmed feed rate.....	443
Interpretation of the programmed rotary axis coordinates.....	443
Type of interpolation between the starting and end position.....	445
Resetting the TCPM FUNCTION.....	446



12.6 Three-dimensional tool compensation (software option 2)..... 447

Introduction.....447
Definition of a normalized vector..... 448
Permitted tool shapes..... 449
Using other tools: Delta values..... 449
3-D compensation without TCPM..... 449
Face Milling: 3D compensation with TCPM..... 450
Peripheral Milling: 3-D radius compensation with TCPM and radius compensation (RL/RR)..... 451

13 Programming: Pallet editor.....	453
13.1 Pallet Management.....	454
Application.....	454
Select pallet table.....	456
Exiting the pallet file.....	456
Run pallet file.....	456



14 Programming: Turning Operations.....	459
14.1 Turning Operations on Milling Machines (Software Option 50).....	460
Introduction.....	460
14.2 Basis Functions (Software Option 50).....	461
Switching between milling/turning mode of operation.....	461
Graphical display of turning operations.....	463
Program spindle speed.....	464
Feed rate.....	465
Tool call.....	465
Tool compensation in the program.....	466
Tool data.....	467
Tool tip radius compensation TRC.....	472
Recessing and undercutting.....	473
Inclined turning.....	479
14.3 Unbalance Functions.....	481
Unbalance while turning.....	481
Measure Unbalance cycle.....	483

15 Manual operation and setup.....	485
15.1 Switch-on, switch-off.....	486
Switch-on.....	486
Switch-off.....	488
15.2 Moving the machine axes.....	489
Note.....	489
Moving the axis with the machine axis direction buttons.....	489
Incremental jog positioning.....	489
Traverse with electronic handwheels.....	490
15.3 Spindle speed S, feed rate F and miscellaneous function M.....	500
Application.....	500
Entering values.....	500
Adjusting spindle speed and feed rate.....	501
Activating feed-rate limitation.....	501
15.4 Functional safety FS (option).....	502
Miscellaneous.....	502
Explanation of terms.....	503
Checking the axis positions.....	504
Overview of permitted feed rates and speeds.....	505
Activating feed-rate limitation.....	505
Additional status displays.....	506
15.5 Datum setting without a 3-D touch probe.....	507
Note.....	507
Preparation.....	507
Workpiece presetting with axis keys.....	507
Datum management with the preset table.....	508
15.6 Using 3-D touch probes.....	514
Overview.....	514
Functions in touch probe cycles.....	516
Selecting touch probe cycles.....	518
Recording measured values from the touch-probe cycles.....	519
Writing measured values from the touch probe cycles in a datum table.....	520
Writing measured values from the touch probe cycles in the preset table.....	521



15.7 Calibrating a 3-D touch trigger probe..... 522

Introduction.....522
Calibrating the effective length..... 523
Calibrating the effective radius and compensating center misalignment..... 524
Displaying calibration values..... 526

15.8 Compensating workpiece misalignment with 3-D touch probe..... 527

Introduction.....527
Identifying basic rotation.....528
Saving a basic rotation in the preset table..... 528
Compensation of workpiece misalignment by rotating the table..... 528
Displaying a basic rotation..... 529
Canceling a basic rotation..... 529

15.9 Datum Setting with 3-D Touch Probe..... 530

Overview..... 530
Datum setting in any axis..... 530
Corner as datum..... 531
Circle center as datum.....533
Setting a center line as datum.....535
Measuring workpieces with a 3-D touch probe.....536
Using touch probe functions with mechanical probes or measuring dials..... 539

15.10 Tilting the working plane (software option 1)..... 540

Application, function..... 540
Traversing reference points in tilted axes..... 542
Position display in a tilted system..... 542
Limitations on working with the tilting function..... 542
To activate manual tilting:..... 543
Setting the current tool-axis direction as the active machining direction..... 544
Setting the datum in a tilted coordinate system..... 545

16 Positioning with Manual Data Input.....	547
16.1 Programming and executing simple machining operations.....	548
Positioning with manual data input (MDI).....	548
Protecting and erasing programs in \$MDI.....	551

17 Test run and program run.....	553
17.1 Graphics.....	554
Application.....	554
Speed of the Setting test runs.....	555
Overview: Display modes.....	556
Plan view.....	557
Projection in three planes.....	557
3-D view.....	558
Magnifying details.....	560
Repeating graphic simulation.....	561
Tool display.....	561
Measurement of machining time.....	562
3-D line graphics.....	563
17.2 Showing the workpiece blank in the working space.....	565
Application.....	565
17.3 Functions for program display.....	566
Overview.....	566
17.4 Test Run.....	567
Application.....	567
17.5 Program run.....	570
Application.....	570
Running a part program.....	571
Interrupt machining.....	572
Moving the machine axes during an interruption.....	573
Resuming program run after an interruption.....	573
Any entry into program (mid-program startup).....	575
Returning to the contour.....	577
17.6 Automatic program start.....	578
Application.....	578
17.7 Optional block skip.....	579
Application.....	579
Inserting the "/" character.....	579
Erasing the "/" character.....	579

178	Optional program-run interruption.....	580
	Application.....	580



18 MOD functions.....	581
18.1 MOD function.....	582
Selecting MOD functions.....	582
Changing the settings.....	582
Exiting MOD functions.....	582
Overview of MOD functions.....	583
18.2 Machine settings.....	584
External access.....	584
Select kinematics.....	586
18.3 Position Display Types.....	587
Application.....	587
18.4 Unit of Measurement.....	588
Application.....	588
18.5 Displaying operating times.....	588
Application.....	588
18.6 Software numbers.....	589
Application.....	589
18.7 Entering the code number.....	589
Application.....	589
18.8 Setting up data interfaces.....	590
Serial interfaces on the TNC 640.....	590
Application.....	590
Setting the RS-232 interface.....	590
Setting the BAUD RATE (baudRate).....	590
Setting the protocol (protocol).....	591
Setting data bits (dataBits).....	591
Check parity (parity).....	591
Setting the stop bits (stopBits).....	591
Setting handshaking (flowControl).....	592
File system for file operations (fileSystem).....	592
Settings for data transfer with the TNCserver PC software.....	592
Setting the operating mode of the external device (fileSystem).....	593
Data transfer software.....	594

18.9 Ethernet interface.....	596
Introduction.....	596
Connection options.....	596
Configuring the TNC.....	596
18.10 Configure HR 550 FS wireless handwheel.....	602
Application.....	602
Assigning the handwheel to a specific handwheel holder.....	602
Setting the transmission channel.....	603
Selecting the transmitter power.....	603
Statistical data.....	604



19 Tables and overviews.....	605
19.1 Machine-specific user parameters.....	606
Application.....	606
19.2 Connector pin layout and connection cables for data interfaces.....	616
RS-232-C/V.24 interface for HEIDENHAIN devices.....	616
Non-HEIDENHAIN devices.....	618
Ethernet interface RJ45 socket.....	618
19.3 Technical Information.....	619
19.4 Overview tables.....	627
Fixed cycles.....	627
Miscellaneous functions.....	628
19.5 Functions of the TNC 640 and the iTNC 530 compared.....	630
Comparison: Specifications.....	630
Comparison: Data interfaces.....	630
Comparison: Accessories.....	631
Comparison: PC software.....	631
Comparison: Machine-specific functions.....	632
Comparison: User functions.....	632
Comparison: Cycles.....	639
Comparison: Miscellaneous functions.....	641
Comparison: Touch probe cycles in the Manual Operation and El. Handwheel modes.....	643
Comparison: Touch probe cycles for automatic workpiece inspection.....	644
Comparison: Differences in programming.....	645
Comparison: Differences in Test Run, functionality.....	649
Comparison: Differences in Test Run, operation.....	649
Comparison: Differences in Manual Operation, functionality.....	649
Comparison: Differences in Manual Operation, operation.....	651
Comparison: Differences in Program Run, operation.....	651
Comparison: Differences in Program Run, traverse movements.....	652
Comparison: Differences in MDI operation.....	656
Comparison: Differences in programming station.....	657