HEIDENHAIN
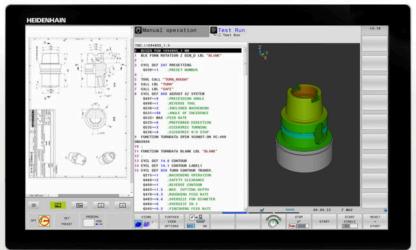
TNC 640

User's Manual
Setup, Testing and Running
NC Programs

NC Software
340590-10
340591-10
340595-10

English (en)
10/2019

Controls and displays

Keys

If you are using a TNC 640 with touch control, you can replace some keystrokes with gestures.

Further information: "Operating the touchscreen", Page 533

Keys on the screen

Key	Function
	Select screen layout
	Toggle the display between machine operating mode, programming mode, and a third desktop
	Soft keys for selecting functions on screen
	Switch the soft-key rows

Alphabetic keyboard

Key	Function
Q W E	File names, comments
G F S	DIN/ISO programming

Machine operating modes

Key	Function
	Manual operation
	Electronic handwheel
	Positioning with Manual Data Input
	Program Run, Single Block
	Program Run, Full Sequence

Programming modes

Key	Function
	Programming
	Test Run

Entering and editing coordinate axes and numbers

Key	Function
X . . . V	Select the coordinate axes or enter them in the NC program
0 . . . 9	Numbers
. −/+	Decimal separator / Reverse algebraic sign
P I	Polar coordinate entry / Incremental values
Q	Q parameter programming / Q parameter status
	Capture actual position
NO ENT	Skip dialog questions, delete words
ENT	Confirm entry and resume dialog
END	Conclude the NC block, end your input
CE	Clear entries or error message
DEL	Abort dialog, delete program section

Tool functions

Key	Function
TOOL DEF	Define tool data in the NC program
TOOL CALL	Call tool data

Managing NC programs and files, control functions

Key	Function
PGM MGT	Select or delete NC programs or files, external data transfer
PGM CALL	Define program call, select datum and point tables
MOD	Select MOD functions
HELP	Display help text for NC error messages, call TNCguide
ERR	Display all current error messages
CALC	Show calculator
SPEC FCT	Show special functions
⊟▶	Currently not assigned

Navigation keys

Key	Function
↑ ←	Position the cursor
GOTO □	Go directly to NC blocks, cycles, and parameter functions
HOME	Navigate to the beginning of a program or table
END	Navigate to the end of the program or table row
PG UP	Navigate up one page
PG DN	Navigate down one page
▤	Select the next tab in forms
⊟↑ ⊟↓	Up/down one dialog box or button

Cycles, subprograms and program section repeats

Key		Function
TOUCH PROBE		Define touch probe cycles
CYCL DEF	CYCL CALL	Define and call cycles
LBL SET	LBL CALL	Enter and call subprograms and program section repeats
STOP		Enter program stop in an NC program

Program path contours

Key	Function
APPR DEP	Contour approach and departure
FK	FK free contour programming
L	Straight line
CC	Circle center/pole for polar coordinates
C	Circular arc with center
CR	Circular arc with radius
CT	Circular arc with tangential transition
CHF RND	Chamfer/rounding arc

Potentiometer for feed rate and spindle speed

Feed rate	Spindle speed

Contents

3.8   Accessories: HEIDENHAIN 3-D touch probes and electronic handwheels....................................127

1

Fundamentals

1.1 About this manual

Safety precautions

Comply with all safety precautions indicated in this document and in your machine tool builder's documentation!

Precautionary statements warn of hazards in handling software and devices and provide information on their prevention. They are classified by hazard severity and divided into the following groups:

⚠ DANGER

Danger indicates hazards for persons. If you do not follow the avoidance instructions, the hazard **will result in death or severe injury.**

⚠ WARNING

Warning indicates hazards for persons. If you do not follow the avoidance instructions, the hazard **could result in death or serious injury**.

⚠ CAUTION

Caution indicates hazards for persons. If you do not follow the avoidance instructions, the hazard **could result in minor or moderate injury.**

NOTICE

Notice indicates danger to material or data. If you do not follow the avoidance instructions, the hazard **could result in things other than personal injury, such as property damage**.

Sequence of information in precautionary statements

All precautionary statements comprise the following four sections:

- Signal word indicating the hazard severity
- Type and source of hazard
- Consequences of ignoring the hazard, e.g.: "There is danger of collision during subsequent machining operations"
- Escape – Hazard prevention measures

Informational notes

Observe the informational notes provided in these instructions to ensure reliable and efficient operation of the software.

In these instructions, you will find the following informational notes:

ⓘ	The information symbol indicates a **tip**. A tip provides additional or supplementary information.

⚙	This symbol prompts you to follow the safety precautions of your machine tool builder. This symbol also indicates machine-dependent functions. Possible hazards for the operator and the machine are described in the machine manual.

📖	The book symbol represents a **cross reference** to external documentation, e.g. the documentation of your machine tool builder or other supplier.

Have you found any errors or would you like to suggest changes?

We are continuously striving to improve our documentation for you. Please help us by sending your suggestions to the following e-mail address:

tnc-userdoc@heidenhain.de

1.2 Control model, software and features

This manual describes functions for setting up the machine as well as for testing and running your NC programs. These functions are provided by controls as of the following NC software numbers.

Control model	NC software number
TNC 640	340590-10
TNC 640 E	340591-10
TNC 640 Programming Station	340595-10

The suffix E indicates the export version of the control. The following software option is unavailable or only available to a limited extent in the export version:

■ Advanced Function Set 2 (option 9) limited to four-axis interpolation

The machine manufacturer adapts the usable features of the control to his machine by the setting machine parameters. Some of the functions described in this manual may therefore not be among the features provided by the control on your machine tool.

Control functions that may not be available on your machine include:

■ Tool measurement with the TT

In order to find out about the actual features of your machine, please contact the machine manufacturer.

Many machine manufacturers, as well as HEIDENHAIN, offer programming courses for the HEIDENHAIN controls. Participation in one of these courses is recommended to familiarize yourself thoroughly with the control's functions.

📖 **Cycle Programming User's Manual:**

All of the cycle functions (touch probe cycles and fixed cycles) are described in the **Cycle Programming** User's Manual. If you need this User's Manual, please contact HEIDENHAIN.
ID: 892905-xx

📖 **User's Manuals for Conversational Programming and ISO Programming:**

All information regarding NC programming (except touch probe cycles and fixed cycles) is provided in the **Conversational Programming** and **ISO Programming** User's Manuals. If you need these User's Manuals, please contact HEIDENHAIN.
ID for Conversational Programming: 892903-xx
ID for ISO Programming: 892909-xx

Software options

The TNC 640 features various software options that can be enabled by your machine tool builder. Each option is to be enabled separately and contains the following respective functions:

Additional Axis (options 0 to 7)

Additional axis	Additional control loops 1 to 8

Advanced Function Set 1 (option 8)

Expanded functions Group 1	**Machining with rotary tables** ■ Cylindrical contours as if in two axes ■ Feed rate in distance per minute **Coordinate conversions:** Tilting the working plane

Advanced Function Set 2 (option 9)

Expanded functions Group 2 Export license required	**3-D machining:** ■ 3-D tool compensation through surface-normal vectors ■ Using the electronic handwheel to change the angle of the swivel head during program run; the position of the tool point remains unchanged (TCPM = Tool Center Point Management) ■ Keeping the tool normal to the contour ■ Tool radius compensation normal to the tool direction ■ Manual traverse in the active tool-axis system **Interpolation:** Linear in > 4 axes (export license required)

HEIDENHAIN DNC (option 18)

	Communication with external PC applications over COM component

Dynamic Collision Monitoring – DCM (option 40)

Dynamic Collision Monitoring	■ The machine manufacturer defines objects to be monitored ■ Warning in Manual operation ■ Collision monitoring in the Test Run mode ■ Program interrupt in Automatic operation ■ Includes monitoring of 5-axis movements

CAD Import (option 42)

CAD import	■ Support for DXF, STEP and IGES ■ Adoption of contours and point patterns ■ Simple and convenient specification of presets ■ Selecting graphical features of contour sections from conversational programs

Adaptive Feed Control – AFC (option 45)

Adaptive Feed Control	**Milling:** ■ Recording the actual spindle power by means of a teach-in cut ■ Defining the limits of automatic feed rate control ■ Fully automatic feed control during program run **Turning (option 50):** ■ Cutting force monitoring during machining

KinematicsOpt (option 48)

Optimizing the machine kinematics	■ Backup/restore active kinematics ■ Test active kinematics ■ Optimize active kinematics

Mill-Turning (option 50)

Milling and turning modes	**Functions:** ■ Switching between Milling/Turning mode of operation ■ Constant surface speed ■ Tool-tip radius compensation ■ Turning cycles ■ Cycle 880: Gear hobbing (option 50 and option 131)

KinematicsComp (option 52)

Three-dimensional compensation	Compensation of position and component errors

OPC UA NC Server 1 - 6 (Options 56 - 61)

Standardized interface	The OPC UA NC Server provides a standardized interface (OPC UA) for external access to the control's data and functions These software options allow you to create up to six parallel client connections

3D-ToolComp (option 92)

3-D tool radius compensation depending on the tool's contact angle Export license required	■ Compensate the deviation of the tool radius depending on the tool's contact angle ■ Compensation values in a separate compensation value table ■ Prerequisite: Working with surface normal vectors (**LN** blocks)

Extended Tool Management (option 93)

Extended tool management	Python-based

Advanced Spindle Interpolation (option 96)

Interpolating spindle	**Interpolation turning:** ■ Cycle 291: Interpolation turning, coupling ■ Cycle 292: Interpolation turning, contour finishing

Spindle Synchronism (option 131)

Spindle synchronization	■ Synchronization of milling spindle and turning spindle ■ Cycle 880: Gear hobbing (option 50 and option 131)

Remote Desktop Manager (option 133)

Remote operation of external computer units	■ Windows on a separate computer unit
	■ Incorporated in the control's interface

Synchronizing Functions (option 135)

Synchronization functions	**Real Time Coupling – RTC:**
	Coupling of axes

Visual Setup Control – VSC (option 136)

Camera-based monitoring of the setup situation	■ Record the setup situation with a HEIDENHAIN camera system
	■ Visual comparison of planned and actual status in the workspace

State Reporting Interface – SRI (option 137)

HTTP accesses to the control status	■ Reading out the times of status changes
	■ Reading out the active NC programs

Cross Talk Compensation – CTC (option 141)

Compensation of axis couplings	■ Determination of dynamically caused position deviation through axis acceleration
	■ Compensation of the TCP (**T**ool **C**enter **P**oint)

Position Adaptive Control – PAC (option 142)

Adaptive position control	■ Changing of the control parameters depending on the position of the axes in the working space
	■ Changing of the control parameters depending on the speed or acceleration of an axis

Load Adaptive Control – LAC (option 143)

Adaptive load control	■ Automatic determination of workpiece weight and frictional forces
	■ Changing of control parameters depending on the actual mass of the workpiece

Active Chatter Control – ACC (option 145)

Active chatter control	Fully automatic function for chatter control during machining

Active Vibration Damping – AVD (option 46)

Active vibration damping	Damping of machine oscillations to improve the workpiece surface

Batch Process Manager (option 154)

Batch process manager	Planning of production orders

Component Monitoring (option 155)

Component monitoring without external sensors	Monitoring configured machine components for overload

Grinding (Option 156)

Jig grinding	■ Reciprocating stroke cycles
	■ Cycles for dressing
	■ Support of the "dressing tool" and "grinding tool" tool types

Gear Cutting (option 157)

Machining gear systems	■ Cycle 285: Define gear wheel
	■ Cycle 286: Gear hobbing
	■ Cycle 287: Gear skiving

Advanced Function Set Turning (option 158)

Advanced turning functions	Cycle 883: Simultaneous turning

Opt. contour milling (Option 167)

Optimized contour cycles	■ Cycle 271: **OCM CONTOUR DATA**
	■ Cycle 272: **OCM ROUGHING**
	■ Cycle 273: **OCM FINISHING FLOOR**
	■ Cycle 274: **OCM FINISHING SIDE**

Feature Content Level (upgrade functions)

Along with software options, significant further improvements of the control software are managed via the **F**eature **C**ontent **L**evel upgrade functions. If you install a software update on your control you do not automatically have the functions available as covered by the FCL.

> **i** All upgrade functions are available to you without surcharge when you receive a new machine.

Upgrade functions are identified in the manual as **FCL n**. The **n** signifies the serial number of the development status.

You can purchase a code number in order to permanently enable the FCL functions. For more information, contact your machine tool builder or HEIDENHAIN.

Intended place of operation

The control complies with the limits for a Class A device in accordance with the specifications in EN 55022, and is intended for use primarily in industrially-zoned areas.

Legal information

This product uses open-source software. Further information is available on the control as follows:

▶ Press the **MOD** key
▶ Select **Code-number entry**
▶ **LICENSE INFO** soft key

New functions 34059x-09

Further information: User's Manual for **Conversational Programming** or **ISO Programming**

- It is now possible to work with cutting data tables.
- The **TCPM** function can also consider spatial angles for Peripheral Milling.
- New **PLANE XY ZX YZ** soft key for selecting the working plane during FK programming.
- In **Test Run** operating mode, a counter defined in the NC program is simulated.
- An NC program you called can be edited when it has been completely executed in the calling NC program.
- In the CAD viewer, you can define the preset or the datum by directly entering the values in the list view window.
- With **TOOL DEF**, you can use QS parameters for entering the data.
- You can now use QS parameters to read from and write to freely definable tables.
- The FN 16 function was extended to include the ***** input character that can be used to write comment lines.
- New output format for the FN 16 function **%RS** that you can use to output texts without formatting.
- The FN 18 functions have been expanded.
- The new user administration enables you to create and administrate users with different access rights, see "User administration", Page 495
- The new **Component Monitoring** software option enables automatic checking of defined machine components for overload, see "Additional status displays", Page 70
- With the new HOST COMPUTER MODE function, you can transfer command to an external host computer, see "Permitting or restricting external access", Page 435
- With the **State Reporting Interface** (**SRI**), HEIDENHAIN provides a simple and reliable interface for acquiring the operating states of your machine, see "State Reporting Interface (option 137)", Page 467
- The basic rotation is taken into account in the **Manual Operation** mode, see "Activating manual tilting:", Page 244
- The new **PROGRAM + MACHINE** screen layout shows you the NC program, collision objects and the workpiece, see "Modes of operation", Page 65
- The new **MACHINE** screen layout shows you the collision objects and the workpiece, see "Modes of operation", Page 65
- The screen layout soft keys were adapted, see "Modes of operation", Page 65
- The additional status display shows the path and angle tolerances without Cycle 32 being active, see "Additional status displays", Page 70
- The additional status display indicates whether the path and angle tolerances are limited by DCM, see "Additional status displays", Page 70

- The control checks all NC programs for completeness before machining. If you attempt to start an incomplete NC program, the control aborts with an error message, see "Data transfer to or from an external data carrier", Page 90.
- In the **Positioning w/ Manual Data Input** operating mode, you can now skip NC blocks, see "Skipping NC blocks", Page 269
- Two new tool types have been added to the tool table: **Ball-nose cutter** and **Toroid cutter**, see "Available tool types", Page 157
- An active TCPM is taken into account during presetting with a 3-D touch probe, see "Presetting with active TCPM", Page 232
- During probing in a plane (Probing PL) you can select the solution when aligning the rotary axes, see "Measuring 3-D basic rotation", Page 228
- The appearance of the **Optional program run stop** has changed, see "Optional program-run interruption", Page 268
- You can use the key between **PGM MGT** and **ERR** to toggle between screens.
- The control supports USB devices with the exFAT file system, see "USB devices on the control", Page 88
- The control can show a handwheel superimposition in the position display even if it was activated using the Global Program Settings (GPS), see "Handwheel superimp.", Page 359
- If the feed rate is less than 10, the control also shows one of the decimal place that have been entered, see "Entering values", Page 186
- In **Test Run** operating mode, the machine tool builder can define whether the tool table or the expanded tool management is opened.
- The machine tool builder defines which file types you will be able to import when using the **ADAPT NC PGM / TABLE** function, see "Importing a file from an iTNC 530", Page 93
- New machine parameter **CfgProgramCheck** (no. 129800) for defining settings for the tool usage files, see "List of user parameter", Page 552

Modified functions 34059x-09

Further information: User's Manual for **Conversational Programming** or **ISO Programming**

- The **PLANE** functions provide the alternative **SYM**selection option in addition to **SEQ**.
- The cutting data calculator has been improved.
- The **CAD-Viewer** now outputs **PLANE SPATIAL** instead of **PLANE VECTOR**.
- The **CAD-Viewer** now outputs 2-D contours by default.
- When programming straight-line blocks, the **&Z** option is no longer shown by default.
- The control does not run a tool change macro if neither a tool name nor a tool number is programmed in the tool call, but the same tool axis as in the previous **TOOL CALL** block.
- The control issues an error message if you combine an FK block with M89.
- When using **SQL UPDATE** and **SQL INSERT**, the control checks the length of the table columns to be written to.
- When using the FN16 function, M_CLOSE and M_TRUNCATE have the same effect as far as output to the screen is concerned.
- The **Batch Process Manager** can now be opened in the **Programming**, **Program run, full sequence** and **Program run, single block** operating modes, see "Batch Process Manager (option 154)", Page 377
- In the **Test Run** operating mode, the **GOTO** key now has the same effect as in the other operating modes, see "GOTO function", Page 276
- If axis angle not equal to tilt angle, the control no longer issues an error message during presetting with manual probing functions, but opens the **Working plane is inconsistent** menu, see "Using a 3-D touch probe ", Page 206
- The **ACTIVATE PRESET** soft key also updates the values of a line activated in the preset management, see "Activating a preset", Page 202
- From the third desktop you can switch to any operating mode using the operating mode keys.
- The additional status display in the **Test Run** operating mode was adapted to match that of the **Manual operation** mode, see "Additional status displays", Page 70
- The control allows updating of the web browser, see "Additional tools for management of external file types", Page 94
- The Remote Desktop Manager allows you to enter an additional waiting time for the shutdown connection, see "Shutting down or rebooting an external computer", Page 453
- The obsolete tool types were removed from the tool table. The **Undefined** type is assigned to any existing tools of these tool types, see "Available tool types", Page 157
- In the expanded tool management, you can now go to the context-sensitive on-line help even while editing the tool form.
- The screensaver glideshow was removed.

- The machine tool builder can specify the axis-specific effect of a shift (mW-CS) of the rotary axes, see "Shift (mW-CS)", Page 356

- The machine tool builder can define the minimum distance between two collision-monitored objects in the **Manual operation** mode.

- The machine tool builder can specify which M functions to allow in the **Manual Operation** mode, see "Application", Page 186

- The machine tool builder can define the default values for the L-OFFS and R-OFFS columns in the tool table, see "Entering tool data into the table", Page 137

New and modified cycle functions 34059x-09
Further information:Cycle Programming User's Manual

■ New Cycle 285 DEFINE GEAR WHEEL (option 157).

■ New Cycle 286 GEAR HOBBING (option 157).

■ New Cycle 287 GEAR SKIVING (option 157).

■ New Cycle 883 TURNING SIMULTANEOUS FINISHING (Option 50 and Option 158).

■ New Cycle 1410 PROBING ON EDGE.

■ New Cycle 1411 PROBING TWO CIRCLES.

■ New Cycle 1420 PROBING IN PLANE.

■ Automatic Touch Probe Cycles 408 to 419 take chkTiltingAxes (no. 204600) into account during presetting.

■ Touch Probe Cycles 41x, automatic preset measurement: New behavior of cycle parameters Q303 MEAS. VALUE TRANSFER and Q305 NUMBER IN TABLE.

■ In Cycle 420 MEASURE ANGLE, the data from the cycle and the touch probe table is taken into account during pre-positioning.

■ Cycle 444 PROBING IN 3-D checks whether the positions of the rotary axes agree with the tilt angles depending on the setting of the optional machine parameter.

■ The help graphic in Cycle 444 PROBING IN 3-D for Q309 ERROR REACTION has been modified and this cycle takes into account a TCPM.

■ Cycle 450 SAVE KINEMATICS does not write the same values during restoring.

■ Cycle 451 MEASURE KINEMATICS was expanded to include value 3 in cycle parameter Q406 MODE.

■ In Cycles 451 MEASURE KINEMATICS and 453 KINEMATICS GRID, the radius of the calibration sphere is only monitored during the second measurement.

■ A simulated touch probe is considered in the simulation. The simulation runs without error message.

■ The REACTION column was added to the touch probe table.

■ In Cycle 24 SIDE FINISHING, a tangential helix is used for approaching and departing in the last infeed.

■ Parameter Q367 SURFACE POSITION was added to Cycle 233 FACE MILLING.

■ Cycle 257 CIRCULAR STUD, now uses Q207 FEED RATE MILLING for roughing, too.

■ The configuration CfgGeoCycle (no. 201000) is taken into account in Cycles 291 COUPLG.TURNG.INTERP. and 292 CONTOUR.TURNG.INTRP..

■ Parameter Q531 ANGLE OF INCIDENCE was extended to 0.001° in Cycle 800 ADJUST XZ SYSTEM.

■ Machine parameter CfgThreadSpindle (no. 113600) is available for use.

New functions 34059x-10

Further information: User's Manual for **Conversational Programming** or **ISO Programming**

- The jig grinding function (Option 156) allows to machine a workpiece using a grinding tool. On the path contour, a superimposed reciprocating stroke is possible.
- The **FUNCTION DRESS** dressing function (Option 156) allows to dress grinding tools.
- Using the compensation tables, the control can compensate for deviations in the tool coordinate system (T-CS) or working plane coordinate system (WPL-CS), also during the program run.
- In the **Batch Process Manager**, the common collision check of all NC programs used for a pallet is available.
- The column order of a table created with the **CREATE TABLE** function corresponds to the order within the **AS SELECT** command.
- With the **FUNCTION TCPM** function, you can define a feed-rate limit of the compensation movements.
- The **FUNCTION TCPM** function is available for ISO programming.
- The control performs a backup of active NC programs to a service file up to a maximum size of 10 MB.
- The FN 18 functions have been extended.
- The machine tool builder uses an optional machine parameter to define the distance to a software limit switch or a collision object for retraction movements.
- The machine tool builder defines in an optional machine parameter whether the control will automatically clear pending warning or error messages when a new NC program is selected or the previous NC program is restarted.
- The **OPC UA NC Server** software options 1 - 6 (Options 51 - 56) provide a standardized interface (OPC UA) for remote access to the control's data and functions, see "HEIDENHAIN OPC UA NC Server (Options 56 - 61)", Page 526
- To simplify the setup of an OPC UA application, the control provides a configuration assistant as a HEROS function, see "Setting up the connection", Page 527
- In the default scope of delivery, the control provides high resolution of the display steps without the **Display Step** (Option 23) software option.
- Additional tool types are available for the definition of grinding tools and dressing tools.
- The **TOOL** tab in the additional status display shows specific data for grinding tools and dressing tools.
- Extended Tool Management also allows to apply the current position value as the tool length.
- The general status display shows an active tool radius compensation using various icons, see "General status display", Page 68
- With the **ACTIVATE AUTOMATIC SAVING** soft key, it is possible to define an error number whose occurrence automatically causes the creation of a service file, see "ACTIVATE AUTOMATIC SAVING soft key", Page 105
- You can take over the position values, axis by axis, to a datum table in the **Program Run Single Block** and **Program Run Full Sequence** operating modes.

- The **POS HR** tab of the additional status display indicates whether the defined Max. val. from **M118** or from the **Global Program Settings** function is applied, see "Additional status displays", Page 70
- In the **BLANK IN WORK SPACE** function, the **RESET REFERENCE POINT** soft key sets the principal axis values of the current preset to 0, see "Showing the workpiece blank in the working space ", Page 265
- In the **BLANK IN WORK SPACE** function, the **Load machine status** soft key is available, see "Showing the workpiece blank in the working space ", Page 265
- The control uses the current preset of the **Test Run** operating mode for simulation, see "Test run execution", Page 273
- The **ADOPT** menu displays either the defined axis angles or the spatial angles, see "Activating manual tilting:", Page 244
- While manual probing functions are active, the control temporarily disables the **Global Program Settings** function.
- In the **Global settings** function, the **ACTIVATE GLOBAL SETTINGS** soft key allows you to restore the settings that were last active, see "Activating and deactivating a function", Page 348

   In the file manager, the **ADVANCED ACCESS RIGHTS** soft key allows you to assign file-specific access rights, see "'public' directory", Page 522

   In addition to the position value, the wireless HR 550 FS handwheel displays values such as the handwheel offset, see "Traverse with electronic handwheels", Page 175
- The control supports the defined traverse limits also for modulo axes, see "Entering traverse limits", Page 433
- With the optional machine parameter **applyCfgLanguage** (no. 101305), you define the behavior of the control if the conversational language in the machine parameters and in the HEROS operating system do not match, see "List of user parameter", Page 552
- In the machine parameter **restoreAxis** (no. 200305), the machine tool builder defines the axis order for returning to the contour in turning mode.
- The machine tool builder specifies the default values to be used by the control for each column of a new preset table line, see "Preset management", Page 194

Functions changed 34059x-10

Further information: User's Manual for **Conversational Programming** or **ISO Programming**

- The control includes the QR parameters in the backup.
- With the **SQL EXECUTE** and **SQL SELECT** SQL commands, it is possible to use composite QS parameters.
- Any display filter you have set in the file manager will remain effective even after a control restart.
- In addition to the **FN 9** step function, it is possible to use the **FN 10** function, which performs a comparison, for QS parameters and texts.

- The control executes the **FN 27: TABWRITE** function and **FUNCTION FILE** in the **Program run, single block** and **Program run, full sequence** operating modes only.
- In machine parameters **fn16DefaultPath** (no. 102202) and **fn16DefaultPathSim** (no. 102203), you can define the path for the output of the **FN 16** function.
- In tool management, the control only displays the input fields related to the selected tool type.
- In the turning tool table, the default value for the **CUTLENGTH** column is 0.
- In the preset table, the input range of columns **SPA**, **SPB**, **SPC**, **A_OFFS**, **B_OFFS**, and **C_OFFS** was extended to +/- 99999.99999.
- On 19" and 24" screens, the control displays up to 10 axes in the additional status display.,see "Additional status displays", Page 70
- Besides other information, the measuring function of the **Test Run** operating mode displays information on the tool, see "Measuring", Page 267
- If user administration is active, the **Retraction after power interruption** function requires the **NC.OPModeManual** right, see "Definition of roles", Page 510
- If user administration is active, the **Global Program Settings** function requires the **NC.OPModeMDI** right, see "Definition of roles", Page 510
- In the additional status display, the **CM** and **CM Detail** tabs are replaced with the **MON** and **MON Detail** tabs.
- When capturing the **Program run:** machine times, the control only considers the active machining status. It is shown by the green **NC Start** icon in the status display.
- Remote access is identified by a new icon, see "Permitting or restricting external access", Page 435
- On handwheels that have a display, the smallest speed level that can be set is 1/1000 of the maximum handwheel speed, see "Traverse with electronic handwheels", Page 175

New and changed cycle functions 34059x-10

Further information:Cycle Programming User's Manual

- New point pattern cycle 224 DATAMATRIX CODE PATTERN for the creation of a DataMatrix code.
- New cycle 238 MEASURE MACHINE STATUS for monitoring machine components for wear.
- New cycle 271 OCM CONTOUR DATA for defining machining information for the OCM cycles.
- New cycle 272 OCM ROUGHING for machining open pockets while maintaining the tool angle.
- New cycle 273 OCM FINISHING FLOOR for machining open pockets while maintaining the tool angle.
- New cycle 274 OCM FINISHING SIDE for machining open pockets while maintaining the tool angle.
- New cycles 1000 DEFINE RECIP. STROKE, 1001 START RECIP. STROKE, and 1002 STOP RECIP. STROKE for grinding with a reciprocation movement.

- New cycles 1010 DRESSING DIAMETER and 1015 PROFILABRICHTEN for dressing a grinding tool.

- New cycle 1030 ACTIVATE WHEEL EDGE for activating the wheel edges.

- New cycles 1032 GRINDING WHL LENGTH COMPENSATION and 1033 GRINDING WHL RADIUS COMPENSATION for compensating the length and radius of a grinding tool.

- New DATUM TABLE soft key in the **Program Run Single Block** and **Program Run Full Sequence** operating modes.

- In cycles 205 UNIVERSAL PECKING and 241 SINGLE-LIP D.H.DRLNG, the input value for Q379 STARTING POINT is checked and compared to Q201 DEPTH.

- Using cycle 225 ENGRAVING, it is possible to engrave the path or name of an NC program.

- If a limit has been programmed in cycle 233, the FACE MILLING cycle will extend the contour in the infeed direction by the corner radius.

- Cycle 239 ASCERTAIN THE LOAD is only displayed if this has been defined by the machine tool builder.

- The help graphics for Q224 ANGLE OF ROTATION in cycle 256 RECTANGULAR STUD was changed.

- The help graphics for Q326 SPACING IN 1ST AXIS and Q327 SPACING IN 2ND AXIS in cycle 415 PRESET INSIDE CORNER was changed.

- Cycle 444 PROBING IN 3-D logs the measured 3-D distance. Thus, the control can distinguish between scrap and rework.

- The help graphics for Q341 PROBING THE TEETH in cycles 481 and 31 CAL. TOOL LENGTH and in cycles 482 and 32 CAL. TOOL RADIUS was changed.

- In cycles 14xx, it is possible to use a handwheel for pre-positioning in semi-automatic mode. After probing, you can move to clearance height manually.

2

First steps

2.1 Overview

This chapter is intended to help you quickly learn to handle the most important procedures on the control. For more information on the respective topic, see the section referred to in the text.

The following topics are included in this chapter:

- Switching on the machine
- Graphically testing the workpiece
- Setting up tools
- Workpiece setup
- Machining the workpiece

> 📖 The User's Manuals for Conversational Programming and ISO Programming cover the following topics:
>
> - Switching on the machine
> - Programming the workpiece

2.2 Switching on the machine

Acknowledging the power interruption and moving to the reference points

> ⚠ **DANGER**
>
> **Caution: Danger for the operator!**
>
> Machines and machine components always present mechanical hazards. Electric, magnetic or electromagnetic fields are particularly hazardous for persons with cardiac pacemakers or implants. The hazard starts when the machine is powered up!
>
> ▶ Read and follow the machine manual
> ▶ Read and follow the safety precautions and safety symbols
> ▶ Use the safety devices

> ⚙ Refer to your machine manual!
> Switching on the machine and traversing the reference points can vary depending on the machine tool.

To switch on the machine, proceed as follows:

▶ Switch on the power supply for the control and the machine
> The control starts the operating system. This process may take several minutes.
> The control will then display the "Power interrupted" message in the screen header.

CE
▶ Press the **CE** key
> The control compiles the PLC program.

I
▶ Switch on the machine control voltage
> The control checks operation of the emergency stop circuit and goes into Reference Run mode.

▶ Cross the reference point manually in the prescribed sequence: For each axis press the **START** key. If you have absolute linear and angle encoders on your machine there is no need for a reference run
> The control is now ready for operation in the **Manual operation** mode.
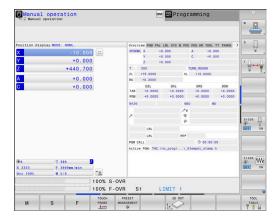
Further information on this topic

- Approaching reference points
  Further information: "Switch-on", Page 168
- Operating modes
  Further information: "Programming", Page 66

2.3 Graphically testing the workpiece

Selecting the Test Run operating mode

You can test NC programs in the **Test Run** operating mode:

⇥
- ▶ Press the operating mode key
- > The control switches to the **Test Run** operating mode.
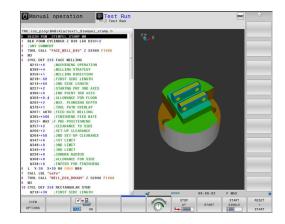
Further information on this topic

- Operating modes of the control
  Further information: "Modes of operation", Page 65
- Testing NC programs
  Further information: "Test run", Page 271

Selecting the tool table

If you have not yet activated a tool table in **Test Run** mode, then you must carry out this step.

[PGM MGT]
- ▶ Press the **PGM MGT** key
- > The control opens the file manager.

[SELECT TYPE]
- ▶ Press the **SELECT TYPE** soft key
- > The control shows a soft-key menu for selection of the file type to be displayed.

[DEFAULT]
- ▶ Press the **DEFAULT** soft key
- > The control shows all saved files in the right-hand window.

[←]
- ▶ Move the cursor to the left onto the directories

[↑]
- ▶ Move the cursor to the **TNC:\table** directory

[→]
- ▶ Move the cursor to the right onto the files

[↓]
- ▶ Position the cursor on the TOOL.T file (active tool table)

[ENT]
- ▶ Accept with the **ENT** key
- > TOOL.T is assigned the status **S** and is thus active for the **Test Run**.

[END]
- ▶ Press the **END** key to close the file manager

Further information on this topic

- Tool management
  Further information: "Entering tool data into the table", Page 137
- Testing NC programs
  Further information: "Test run", Page 271

Selecting the NC program

PGM MGT	▶ Press the **PGM MGT** key
	> The control opens the file manager.
LAST FILES	▶ Press the **LAST FILES** soft key
	> The control opens a pop-up window with the most recently selected files.
	▶ Use the arrow keys to select the NC program that you want to test.
ENT	▶ Accept with the **ENT** key

Selecting the screen layout and the view

⟳	▶ Press the **Screen layout** key
	> The control displays all available alternatives in the soft-key row.
PROGRAM + MACHINE	▶ Press the **PROGRAM + MACHINE** soft key
	> In the left half of the screen the control shows the NC program; in the right half it shows the workpiece blank.

The control provides the following views:

Soft keys	Function
VIEW	Plan view
VIEW	Projection in three planes
VIEW	3-D view

Further information on this topic

- Graphic functions
  Further information: "Graphics ", Page 256
- Performing a test run
  Further information: "Test run", Page 271

Starting the test run

RESET + START

▶ Press the **RESET + START** soft key
> The control resets the previously active tool data.
> The control simulates the active NC program up to a programmed interruption or to the program end.
▶ While the simulation is running, you can use the soft keys to change views

STOP

▶ Press the **STOP** soft key
> The control interrupts the test run.

START

▶ Press the **START** soft key
> The control resumes the test run after an interruption.

Further information on this topic

■ Performing a test run
   Further information: "Test run", Page 271
■ Graphic functions
   Further information: "Graphics ", Page 256
■ Adjusting the simulation speed
   Further information: "Setting the speed of the Test Run mode ", Page 262

2.4 Setting up tools

Selecting the Manual operation mode
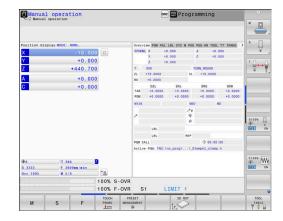
Tools are set up in the **Manual operation** mode:

▶ Press the operating mode key

> The control switches to the **Manual operation** mode.

Further information on this topic

- Operating modes of the control
  Further information: "Modes of operation", Page 65

Preparing and measuring tools

▶ Clamp the required tools in their tool holders

▶ When measuring with an external tool presetter: Measure the tools, note down the length and radius, or transfer them directly to the machine through a transfer program

▶ When measuring on the machine: store the tools in the tool changer
Further information: "Editing the TOOL_P.TCH pocket table", Page 53

Editing the TOOL.T tool table

⚙	Refer to your machine manual! The procedure for calling the tool management may differ from that described below.

In the TOOL.T tool table (permanently stored under **TNC:\table**), you can save tool data such as length and radius, as well as further tool-specific information that the control needs in order to execute a wide variety of functions.

To enter tool data in the tool table TOOL.T, proceed as follows:

▶ Press the **TOOL TABLE** soft key

> The control shows the tool table.

▶ Set the **EDIT** soft key to **ON**

▶ With the upward or downward arrow keys you can select the tool number that you want to edit

▶ With the rightward or leftward arrow keys you can select the tool data that you want to edit

▶ Press the **END** key

> The control closes the tool table and saves the changes.

Further information on this topic

- Operating modes of the control
  Further information: "Modes of operation", Page 65
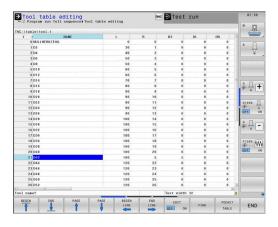
- Working with the tool table
  Further information: "Entering tool data into the table", Page 137

- Using the tool management (option 93)
  Further information: "Calling tool management", Page 153

Editing the TOOL_P.TCH pocket table

⚙ Refer to your machine manual!
    The function of the pocket table depends on the
    machine.

In the pocket table TOOL_P.TCH (permanently saved under
TNC:\table) you specify which tools your tool magazine contains.
To enter data in the pocket table TOOL_P.TCH, proceed as follows:

TOOL TABLE	▶ Press the **TOOL TABLE** soft key
	> The control shows the tool table.
POCKET TABLE	▶ Press the **POCKET TABLE** soft key
	> The control shows the pocket table.
EDIT OFF ON	▶ Set the **EDIT** soft key to **ON**
	▶ With the upward or downward arrow keys you can select the pocket number that you want to edit
	▶ With the rightward or leftward arrow keys you can select the data that you want to edit
END	▶ Press the **END** key

Further information on this topic
- Operating modes of the control
  Further information: "Modes of operation", Page 65
- Working with the pocket table
  Further information: "Pocket table for tool changer",
  Page 145
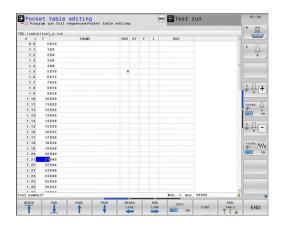
2.5   Workpiece setup

Selecting the correct operating mode

Workpieces are set up in the **Manual operation** or **Electronic handwheel** operating mode

▶ Press the operating mode key

> The control switches to the **Manual operation** mode.

Further information on this topic

- The **Manual operation** mode
  Further information: "Moving the machine axes", Page 173

Clamping the workpiece

Mount the workpiece with a fixture on the machine table. If you have a 3-D touch probe on your machine, then you do not need to clamp the workpiece parallel to the axes.

If you do not have a 3-D touch probe available, you have to align the workpiece so that it is fixed with its edges parallel to the machine axes.

Further information on this topic

- Presetting with a 3-D touch probe
  Further information: "Presetting with a 3-D touch probe ", Page 232
- Presetting without 3-D touch probe
  Further information: "Presetting without a 3-D touch probe", Page 203

Presetting with a 3-D touch probe

Inserting a 3-D touch probe

▶ Select **Positioning w/ Manual Data Input** operating mode

▶ Press the **TOOL CALL** key

▶ Enter the tool data

▶ Press the **ENT** key

▶ Enter the tool axis **Z**

▶ Press the **ENT** key

▶ Press the **END** key

Set a preset

▶ Select **Manual operation** mode

▶ Press the **TOUCH PROBE** soft key

> The control displays the available functions in the soft-key row.

▶ Set the preset at a workpiece corner, for example

▶ Use the axis direction keys to position the touch probe near the first touch point on the first workpiece edge

▶ Select the probing direction via soft key

▶ Press the **NC start** key

> The touch probe moves in the defined direction until it contacts the workpiece and then automatically returns to its starting point.

▶ Use the axis direction keys to pre-position the touch probe near the second touch point on the first workpiece edge

▶ Press the **NC start** key

> The touch probe moves in the defined direction until it contacts the workpiece and then automatically returns to its starting point.

▶ Use the axis direction keys to pre-position the touch probe near the first touch point on the second workpiece edge

▶ Select the probing direction via soft key

▶ Press the **NC start** key

> The touch probe moves in the defined direction until it contacts the workpiece and then automatically returns to its starting point.

▶ Use the axis direction keys to pre-position the touch probe near the second touch point on the second workpiece edge

▶ Press the **NC start** key

> The touch probe moves in the defined direction until it contacts the workpiece and then automatically returns to its starting point.

> The control then displays the coordinates of the measured corner point.

▶ Set to 0: Press the **SET PRESET** soft key

▶ Press the **END** soft key to close the menu

Further information on this topic

■ Presetting
  Further information: "Presetting with a 3-D touch probe ",
  Page 232

2.6 Machining the workpiece

Selecting the Program run, single block or Program run, full sequence operating mode

You can run NC programs either in the **Program run, single block** or the **Program run, full sequence** operating mode:

▶ Press the operating mode key

> The control switches to the **Program run, single block** operating mode and runs the NC program block by block.

▶ You have to confirm each NC block with the **NC start** key

▶ Press the **Program run, full sequence** key

> The control switches to the **Program run, full sequence** operating mode and runs the NC program after NC start up to a program interruption or to the end of the program.

Further information on this topic

■ Operating modes of the control
Further information: "Modes of operation", Page 65

■ Running NC programs
Further information: "Program run", Page 278

Selecting the NC program

▶ Press the **PGM MGT** key

> The control opens the file manager.

▶ Press the **LAST FILES** soft key

> The control opens a pop-up window with the most recently selected files.

▶ If required, use the arrow keys to select the NC program you want to run. Load with the **ENT** key

Starting an NC program

▶ Press the **NC start** key
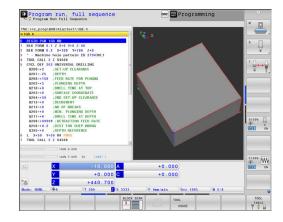
> The control runs the active NC program.

Further information on this topic

■ Running NC programs
Further information: "Program run", Page 278

3

Fundamentals

3.1 The TNC 640

HEIDENHAIN TNC controls are workshop-oriented contouring controls that enable you to program conventional milling and drilling operations right at the machine in easy-to-use Klartext conversational language. They are designed for milling, drilling, and boring machines, as well as for machining centers, with up to 24 axes. You can also change the angular position of the spindle under program control.

An integrated hard disk provides storage for as many NC programs as you like, even if they were created off-line. For quick calculations you can call up the on-screen calculator at any time.

Keyboard and screen layout are clearly arranged in such a way that the functions are fast and easy to use.

HEIDENHAIN Klartext and DIN/ISO

HEIDENHAIN Klartext, the dialog-guided programming language for workshops, is an especially easy method of writing programs. Programming graphics illustrate the individual machining steps for programming the contour. If no NC-dimensioned drawing is available, then the FK free contour programming will help. Workpiece machining can be graphically simulated either during a test run or during a program run.

It is also possible to program in ISO format.

You can also enter and test one NC program while another NC program is machining a workpiece.

Further information: User's Manual for Conversational Programming or ISO Programming

Compatibility

NC programs created on HEIDENHAIN contouring controls (starting from the TNC 150 B) may not always run on the TNC 640. If the NC blocks contain invalid elements, the control will mark these as ERROR blocks or with error messages when the file is opened.

> **i**   Please also note the detailed description of the differences between the iTNC 530 and the TNC 640.
> **Further information:** "Differences between the TNC 640 and the iTNC 530", Page 577

Data security and data protection

The availability of data as well as their guaranteed confidentiality, integrity, and authenticity are decisive for your company's success. HEIDENHAIN therefore places the utmost importance on the protection of relevant data against loss, manipulation, and unauthorized publication.

To ensure that your data is protected actively on the control, HEIDENHAIN offers integrated, state-of-the-art software solutions.

Your control supports the following software solutions:
- SELinux
  Further information: "SELinux security software", Page 494
- Firewall
  Further information: "Firewall", Page 476
- Sandbox
  Further information: "Sandbox tab", Page 491
- Integrated browser
  Further information: "Displaying Internet files", Page 97
- Management of external access
  Further information: "Permitting or restricting external access", Page 435
- Monitoring of TCP and UDP ports
  Further information: "Portscan", Page 462
- Remote diagnosis
  Further information: "Remote Service", Page 463
- User administration
  Further information: "User administration", Page 495

These solutions protect the control effectively, but cannot replace company-specific IT security and an integrated overall strategy. In addition to the solutions provided, HEIDENHAIN recommends implementing a company-specific security strategy. This way you ensure effective protection of your data and information, even after they have been exported from the control.

To ensure data security also in the future, HEIDENHAIN recommends informing yourself regularly about product updates and updating the software to the current version.

⚠ DANGER

Caution: Danger for the operator!

Manipulated data records or software can lead to an unexpected behavior of the machine. Malicious software (viruses, Trojans, malware, or worms) can cause changes to data records and software.

▶ Check any removable memory media for malicious software before using them
▶ Start the internal web browser only from within the sandbox

---

Virus scanners

HEIDENHAIN found that virus scanners can negatively affect the behavior of the NC control.

Possible negative effects are feed rate drops or system crashes, for example. Such negative effects are unacceptable in numerical controls for machine tools. HEIDENHAIN therefore does not offer a virus scanner for the control and does not recommend using a virus scanner.

The control provides the following alternatives:

- SELinux
- Firewall
- Sandbox
- Locking from external access
- Monitoring of TCP and UDP ports

If properly configured, these options ensure extremely effective protection for the data of the control.

If you insist on using a virus scanner, you must operate the control in a protected network (with a gateway and a virus scanner). The subsequent installation of a virus scanner is not possible.

3.2 Visual display unit and operating panel

Display screen

The control is shipped with a 19-inch screen.

1 Header

When the control is on, the screen displays the selected operating modes in the header: The machine operating mode at left and the programming mode at right. The currently active mode is displayed in the larger field of the header, where the dialog prompts and messages also appear (exception: if the control only displays graphics).

2 Soft keys

In the footer the control indicates additional functions in a soft-key row. You can select these functions by pressing the keys immediately below them. The thin bars immediately above the soft-key row indicate the number of soft-key rows that can be called with the keys to the right and left that are used to switch the soft keys. The bar representing the active soft-key row is blue

3 Soft-key selection keys

4 Keys for switching the soft keys

5 Setting the screen layout

6 Key for switchover between machine operating modes, programming modes, and a third desktop

7 Soft-key selection keys for machine tool builders

8 Keys for switching the soft keys for machine tool builders

> ℹ If you are using a TNC 640 with touch control, you can replace some keystrokes with gestures.
> **Further information:** "Operating the touchscreen", Page 533

Setting the screen layout

You select the screen layout yourself. In the **Programming** operating mode, for example, you can have the control show the NC program blocks in the left window while the right window displays programming graphics. You could also display the program structure in the right window instead, or display only the NC program blocks in one large window. The available screen windows depend on the selected operating mode.
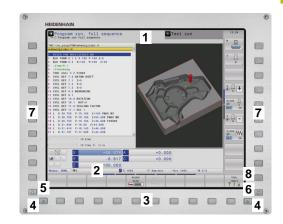
Setting the screen layout:

▶ Press the **screen layout** key: The soft-key row shows the available layout options
Further information: "Modes of operation", Page 65

▶ Select the desired screen layout with a soft key

PROGRAM + GRAPHICS

Control panel

The TNC 640 is delivered with an integrated operating panel. The figure to the right shows the operating elements of the operating panel:

1 Alphabetic keyboard for entering texts and file names, as well as for ISO programming

2 ■ File management
   ■ Calculator
   ■ MOD function
   ■ HELP function
   ■ Show error messages
   ■ Toggle between the operating modes

3 Programming modes

4 Machine operating modes

5 Initiating programming dialogs

6 Navigation keys and **GOTO** jump command

7 Numerical input and axis selection

8 Touchpad

9 Mouse buttons

10 USB connection

The functions of the individual keys are described on the inside front cover.

> **ℹ** If you are using a TNC 640 with touch control, you can replace some keystrokes with gestures.
> **Further information:** "Operating the touchscreen", Page 533

> **⚙** Refer to your machine manual!
> Some machine tool builders do not use the standard HEIDENHAIN operating panel.
> External keys, e.g.**NC START** or **NC STOP**, are described in your machine manual.

Extended Workspace Compact

In widescreen format, the MC 8562 provides additional screen workspace to the left of the control's user interface.

The layout providing the additional screen workspace is called **Extended Workspace Compact**.

This layout enables you to open further applications in addition to the control's user interface so that you can simultaneously keep an eye on the machining process.

The additional screen workspace in **Extended Workspace Compact** mode provides full multitouch support. When you switch to full-screen mode, you can use the HEIDENHAIN keyboard for your external applications.

One **Extended Workspace Compact** area is reserved for the machine tool builder's applications.

Extended Workspace Compact allows you to choose between the following views:

- Screen split into additional screen workspace and main screen
- Full-screen mode of control screen

> **ℹ** HEIDENHAIN also continues offering a second screen for the control as **Extended Workspace Comfort**.

Extended Workspace Compact is divided into three areas:

1 **JH Standard**:

The control's main screen is shown in this area. This area accommodates the control with all its functions.

2 **JH Extended**:

This area provides configurable quick accesses to HEIDENHAIN applications.

Contents of **JH Extended:**

- **HEROS** menu
- 1st screen workspace, **Manual Operation** mode
- 2nd screen workspace, **Programming** operating mode
- 3rd and 4th screen workspaces, freely usable for applications, such as the **CAD Converter**
- Collection of frequently used soft keys
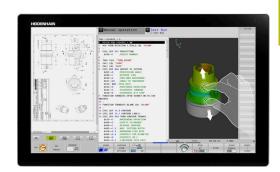
> **ℹ** Benefits of **JH Extended:**
> - Each operating mode has its own additional soft-key row
> - Navigation through the various rows of HEIDENHAIN soft keys is no longer necessary

3 **OEM**:

This area is reserved for the machine tool builder's applications.

Contents of the **OEM** area:

- The machine tool builder can use this area to display functions for Python applications
- This area allows integration of Windows computers into the network

  Further information: "Remote Desktop Manager (option 133)", Page 448

ℹ With the **Remote Desktop Manager** option, you can start additional applications —such as a Windows computer—on your control and have your control display them in the additional screen workspace or in full-screen mode of **Extended Workspace Compact**.

Further information: "Remote Desktop Manager (option 133)", Page 448

In machine parameter **CfgSideScreen** (no. 130000), you can select the connection to be embedded in the second workspace on the screen.

The machine tool builder needs to activate this machine parameter and configure it such that it can be enabled.

In **connection**, you enter the name of the connection defined in the **Remote Desktop Manager** (e.g. Windows 10).

3.3 Modes of operation
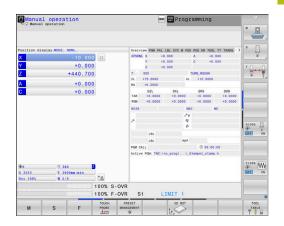
Manual Operation and El. Handwheel

The **Manual operation** mode is required for setting up the machine tool. In this mode of operation, you can position the machine axes manually or by increments, set the presets and tilt the working plane.

The **Electronic handwheel** operating mode supports manual traverse of machine axes with the HR electronic handwheel.

Soft keys for the screen layout (select as described above)

Soft key	Window
POSITION	Positions
POSITION + STATUS	Left: positions, right: status display
POSITION + WORKPIECE	Left: positions, right: workpiece
POSITION + MACHINE	Left: positions, right: collision objects and workpiece
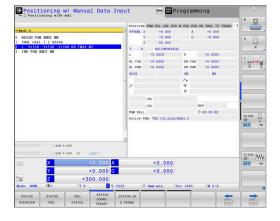
Positioning with Manual Data Input

This mode of operation is used for programming simple traversing movements, such as for face milling or pre-positioning.

Soft keys for selecting the screen layout

Soft key	Window
PGM	NC program
PROGRAM + STATUS	Left: NC program, right: status display
PROGRAM + WORKPIECE	Left: NC program, right: workpiece
PROGRAM + MACHINE	Left: NC program, right: collision objects and workpiece
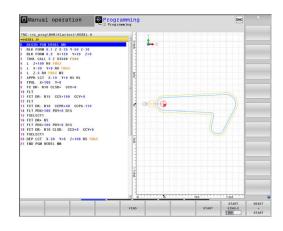
Programming

In this mode of operation you create NC programs. The FK free programming feature, the various cycles and the Q parameter functions help you with programming and add necessary information. If desired, you can have the programming graphics show the programmed paths of traverse.

Soft keys for selecting the screen layout

Soft key	Window
PGM	NC program
PROGRAM + SECTS	Left: NC program, right: program structure
PROGRAM + GRAPHICS	Left: NC program, right: programming graphics
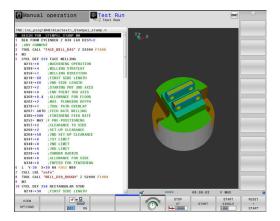
Test Run

In the **Test Run** operating mode, the control checks NC programs and program sections for errors, such as geometrical incompatibilities, missing or incorrect data within the NC program or violations of the working space. This simulation is supported graphically in different display modes.

Soft keys for selecting the screen layout

Soft key	Window
PGM	NC program
PROGRAM + STATUS	Left: NC program, right: status display
PROGRAM + WORKPIECE	Left: NC program, right: workpiece
WORKPIECE	Workpiece
PROGRAM + MACHINE	Left: NC program, right: collision objects and workpiece
MACHINE	Collision objects and workpiece

Program Run, Full Sequence and Program Run, Single Block

In the **Program Run Full Sequence** operating mode, the control runs an NC program continuously to its end or to a manual or programmed stop. You can resume program run after an interruption.
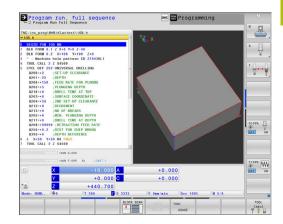
In the **Program Run Single Block** operating mode, you execute each NC block separately by pressing the **NC start** key. With point pattern cycles and **CYCL CALL PAT**, the control stops after each point.

Soft keys for selecting the screen layout

Soft key	Window
PGM	NC program
PROGRAM + SECTS	Left: NC program, right: structure
PROGRAM + STATUS	Left: NC program, right: status display
PROGRAM + WORKPIECE	Left: NC program, right: workpiece
WORKPIECE	Workpiece
POSITION + MACHINE	Left: NC program, right: collision objects and workpiece
MACHINE	Collision objects and workpiece

Soft keys for screen layout with pallet tables

Soft key	Window
PALLET	Pallet table
PROGRAM + PALLET	Left: NC program, right: pallet table
PALLET + STATUS	Left: pallet table, right: status display
PALLET + GRAPHICS	Left: pallet table, right: graphics
BPM	Batch Process Manager

3.4   Status displays

General status display

The general status display in the lower part of the screen informs you of the current state of the machine.

It is displayed automatically in the following operating modes:

- **Program run, single block**
- **Program run, full sequence**
- **Positioning w/ Manual Data Input**
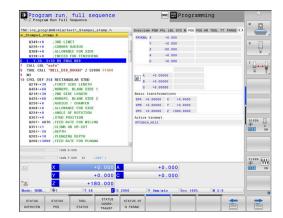
> ℹ️   If the **GRAPHICS** screen layout is selected, the status display is not shown.

In the **Manual operation** and **Electronic handwheel** operating modes, the status display appears in the large window.

Information in the status display

Icon	Meaning
ACTL.	Position display: Actual, nominal or distance-to-go coordinates mode
X Y Z	Machine axes; the control displays auxiliary axes in lower-case letters. The sequence and quantity of displayed axes is determined by the machine tool builder. Refer to your machine manual for more information
⊕	Number of the active preset from the preset table. If the preset was set manually, the control displays the text **MAN** behind the symbol
F S M	The displayed feed rate in inches corresponds to one tenth of the effective value. Spindle speed S, feed rate F and active M functions
✛	Axis is clamped
⊗	Axis can be moved with the handwheel
M-t/9	The spindle is controlled from within a cycle, e.g. during tapping
	RL tool radius compensation is active While the **BLOCK SCAN** function is active, the icon is transparent
	RR tool radius compensation is active While the **BLOCK SCAN** function is active, the icon is transparent
	R+ tool radius compensation is active While the **BLOCK SCAN** function is active, the icon is transparent

Icon	Meaning
	R- tool radius compensation is active
	While the **BLOCK SCAN** function is active, the icon is transparent
	3-D tool compensation is active
	While the **BLOCK SCAN** function is active, the icon is transparent
	A basic rotation is active in the active preset
	Axes are moving, taking the basic rotation into account
	A 3-D basic rotation is active in the active preset
	Axes are moving, taking the setting sin the 3-D ROT menu into account
	Axes are mirrored and moved
TCPM	The **M128** function or **TCPM FUNCTION** is active
	The function for traversing in the tool-axis direction is active
	No NC program selected, NC program reselected, NC program aborted via internal stop, or NC program terminated
	In this condition the control has no modally effective program information (i.e. the contextual reference), so that all handling is possible, e.g. cursor movements or modification of Q parameters.
	NC program started, execution runs
	For safety reasons, the control permits no handling in this condition
	NC program stopped, e.g. in **Program run, full sequence** operating mode after pressing the **NC stop** key
	For safety reasons, the control permits no handling in this condition

Icon	Meaning
	NC program interrupted, e.g. in **Positioning w/ Manual Data Input** operating mode, following the error-free execution of an NC block
	In this condition the control permits various handling, e.g. cursor movements or the modification of Q parameters. With this handling the control may lose the modally effective program information (i.e. the contextual reference). Loss of this contextual reference may cause undesired tool positions!
	Further information: "Positioning w/ Manual Data Input operating mode", Page 306 and "Program-controlled interruptions", Page 283
	NC program aborted or terminated
	Turning mode is active
	Dressing mode is active
	The Dynamic Collision Monitoring function (DCM, Option 40) is active
	The Adaptive Feed Control function (AFC) is active in teach-in cut mode (Option 45)
	The Adaptive Feed Control function (AFC) is active in closed-loop mode (Option 45)
	The Active Chatter Control (ACC) function is active (Option 145)
	Pulsing spindle speed function is active
	Global Program Settings are active (Option 44)

 The order of icons can be changed with the optional machine parameter **iconPrioList** (no. 100813). Only the icons for control-in-operation (STIB) and DCM (option 40) are always visible and cannot be configured.

Additional status displays

The additional status displays contain detailed information on the program run. They can be called in all operating modes. Exception: the **Programming operating mode.** In the **Test Run** operating mode, only a limited status display is available.

To switch on the additional status display

⬛ ▸ Call the soft key row for screen layout

PROGRAM + STATUS ▸ Select the layout option for the additional status display

> In the right half of the screen, the control shows the **Overview** status form.

To select an additional status display

▷ ▸ Toggle through the soft key rows until the **STATUS** soft keys appear

STATUS POS. ▸ Either select the additional status display directly with the soft key, e.g. positions and coordinates; or

⬛ ▸ use the switch-over soft keys to select the desired view

Select the status displays described below as follows:
- directly with the corresponding soft key
- via the switchover soft keys
- or by using the **next tab** key
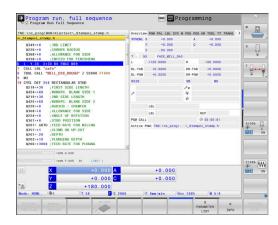
⚙ Please note that some of the status information described below is not available unless the associated software option is enabled on your control.

Overview
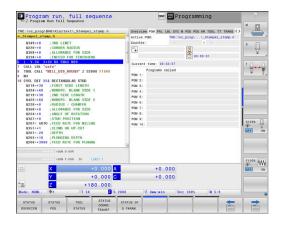
The control displays the **Overview** status form after switch-on if you selected the **PROGRAM + STATUS** (or **POSITION + STATUS**) screen layout. The overview form contains a summary of the most important status information, which you can also find on the various detail forms.

Soft key	Meaning
STATUS OVERVIEW	Position display
	Tool information
	Active M functions
	Active coordinate transformations
	Active subprogram
	Active program section repeat
	NC program called with **PGM CALL**
	Current machining time
	Name and path of the active main program
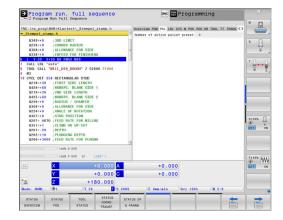
General program information (PGM tab)

Soft key	Meaning
No direct selection possible	Name and path of the active main program
	Actual/nominal value counter
	Circle center CC (pole)
	Dwell time counter
	Current machining time
	Current time
	Called NC programs
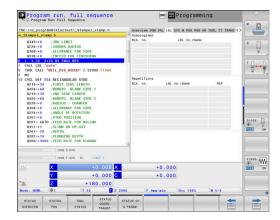
Pallet information (PAL tab)

Soft key	Meaning
No direct selection possible	Number of the active pallet preset

Program section repeats and subprograms (LBL tab)

Soft key	Meaning
No direct selection possible	Active program section repeats with block number, label number, and number of programmed repeats/repeats yet to be run
	Active subprograms with block number in which the subprogram was called and the label number that was called

Information on standard cycles (CYC tab)

Soft key	Meaning
No direct selection possible	Active fixed cycle
	Active path and angle tolerances
	Depending on which path and angle tolerances are active, the following values are displayed:
	■ Values of Cycle 32, Tolerance
	■ Values defined by the machine tool builder
	■ Values limited by DCM
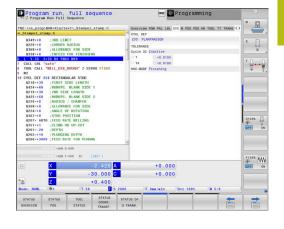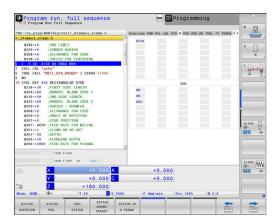
> ⚙ The tolerance limits imposed by DCM are configured by the machine tool builder.
>
> If a tolerance is limited by DCM, the control displays a gray warning triangle as well as the limited values.

Active miscellaneous functions M (M tab)
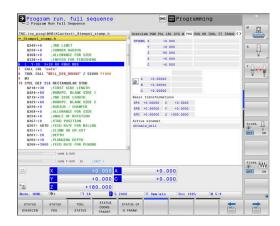
Soft key	Meaning
No direct selection possible	List of the active M functions with fixed meaning
	List of the active M functions that are adapted by your machine manufacturer

Positions and coordinates (POS tab)
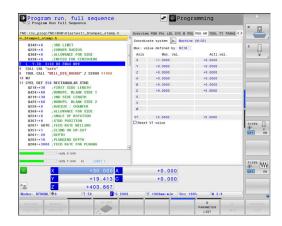
Soft key	Meaning
STATUS POS.	Type of position display, e.g. actual position
	Tilt angle of the working plane
	OEM rotation **Further information:** "Working plane coordinate system WPL-CS", Page 122
	Angle of basic transformations
	Active kinematics

Global Program Settings (POS HR tab)

Soft key	Meaning

No direct selection possible	Current values of **Handwheel superimp.**
	■ Active coordinate system
	   ■ With **M118**: always the machine coordinate system
	   ■ Selectable with **GPS** (Global Program Settings)
	■ Max. val. is defined by **M118** or **GPS**
	■ Max. val. and Actl.val. of the selected axes
	■ Status of the **Reset VT value** function
	Further information: "Global Program Settings (option 44)", Page 346
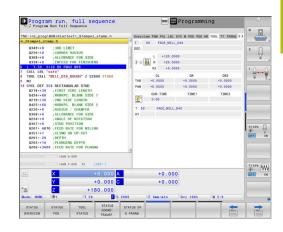
ℹ The values of all other settings provided by the Global Program Settings function are displayed on the **GS** tab.
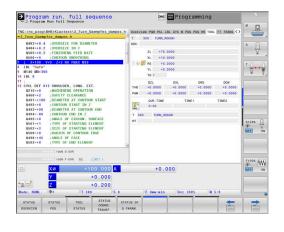
Information on tools (TOOL tab)

Soft key	Meaning
TOOL STATUS	Display of active tool: ■ T: Tool number and tool name ■ RT: Number and name of a replacement tool
	Tool axis
	Tool length and tool radii
	Oversizes (delta values) from the tool table (TAB) and the **TOOL CALL** (PGM)
	Tool life, maximum tool life (TIME 1) and maximum tool life for **TOOL CALL** (TIME 2)
	Display of programmed tool and replacement tool
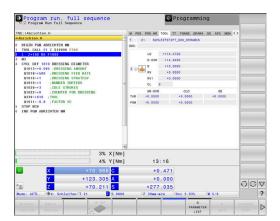
Display for turning tools (TOOL tab)

Soft key	Meaning
TOOL STATUS	Display of active tool: ■ T: Tool number and tool name ■ RT: Number and name of a replacement tool
	Tool axis
	Tool lengths, cutting-edge radius and tool orientation
	Finishing allowances (delta values) from the tool table (TAB) and **FUNCTION TURNDATA CORR** (PGM)
	Tool life, maximum tool life (TIME 1) and maximum tool life for **TOOL CALL** (TIME 2)
	Display of programmed tool and replacement tool

Display for grinding tools (TOOL tab)

Soft key	Meaning
TOOL STATUS	Display of active tool: ■ T: Tool number and tool name ■ DOC: Comment on the tool
	Tool axis
	Tool dimensions and cutting edge (CL: for **Cutter Location**)
	Finishing allowances (delta values) from the tool table (TAB) and NC program (PGM)

Display for dressing tools (TOOL tab)

Soft key	Meaning
TOOL STATUS	Display of active tool: ■ T: Tool number and tool name ■ DOC: Comment on the tool
	Tool axis
	Tool dimensions and tool orientation (TO)
	Finishing allowances (delta values) from the tool table (TAB) and NC program (PGM)
	Tool life
	Display of programmed tool and replacement tool
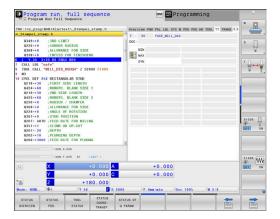
Tool measurement (TT tab)

⚙	The control displays this tab only if the function is active on your machine.
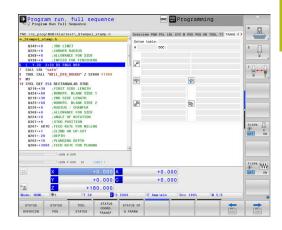
Soft key	Meaning
No direct selection possible	Active tool
	Measured values from tool measurement

Coordinate transformations (TRANS tab)

Soft key	Meaning
STATUS COORD. TRANSF.	Name of the active datum table
	Active datum number (**#**), comment from the active line of the active datum number (**DOC**) from Cycle 7
	Active datum shift (Cycle 7); the control displays an active datum shift in up to 8 axes
	Mirrored axes (Cycle 8)
	Active rotation angle (Cycle 10)
	Active scaling factor/factors (Cycle 11 / 26); the control displays an active scaling factor in up to 6 axes
	Scaling datum

> **i** In the optional machine parameter **CfgDisplayCoordSys** (no. 127501), you can specify the coordinate system in which the status display shows an active datum shift.

Further information: Cycle Programming User's Manual
Further information: User's Manual for Conversational Programming or ISO Programming

Displaying Q parameters (QPARA tab)
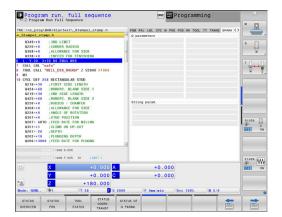
Soft key	Meaning
STATUS OF Q PARAM.	Display the current values of the defined Q parameters
	Display the character strings of the defined string parameters

> **i** Press the **Q PARAMETER LIST** soft key. The control opens a pop-up window. For each parameter type (Q, QL, QR, QS), define the parameter numbers you wish to check. Separate single Q parameters with commas, and connect sequential Q parameters with a hyphen, e.g. 1,3,200-208. The maximum number of characters per parameter type is 132.
>
> The display in the **QPARA** tab always contains eight decimal places. The result of Q1 = COS 89.999 is shown by the control as 0.00001745, for example. Very large and very small values are displayed by the control in exponential notation. The result of Q1 = COS 89.999 * 0.001 is shown by the control as +1.74532925e-08, whereby e-08 corresponds to the factor of 10^{-8}.
>
> The display of QS parameters is restricted to the first 30 characters, i.e. the contents might be truncated.
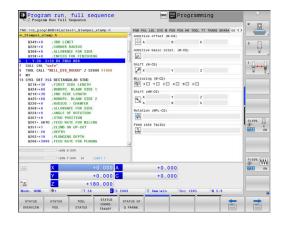
Global Program Settings (GS tab, Option 44)

⚙ The control displays this tab only if the function is active on your machine.

Soft key	Meaning
No direct selection possible	Currently active values of the Global Program Settings function:
	■ **Additive offset (M-CS)**
	■ **Additive basic rotat. (W-CS)**
	■ **Shift (W-CS)**
	■ **Mirroring (W-CS)**
	■ **Shift (mW-CS)**
	■ **Rotation (WPL-CS)**
	■ **Feed rate factor**
	Further information: "Global Program Settings (option 44)", Page 346
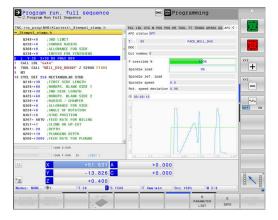
ℹ The control displays the values of the **Handwheel superimp.** setting on the **POS HR** tab.

Adaptive Feed Control (AFC tab, option 45)

⚙ The control displays this tab only if the function is active on your machine.

Soft key	Meaning
No direct selection possible	Active tool (number and name)
	Cut number
	Current factor of the feed potentiometer in %
	Active spindle load in percent
	Reference load of the spindle
	Current spindle speed
	Current deviation of the speed
	Current machining time
	Line diagram, in which the current spindle load and the value commanded by the control for the feed rate override are shown

Monitoring of configured machine components (MON and MON Detail tabs, Option 155)

> ⚙ The control displays these tabs only if the function has been enabled on your machine.
>
> The machine tool builder can define up to ten overload monitoring tasks.
>
> For any detected overload conditions, the machine tool builder has configured component-specific automated responses, e.g. stop of the current execution.

MON tab

Soft key	Meaning
No direct selection possible	**MON status**
	Active if at least one monitoring task has been defined by the machine tool builder
	Monitoring tasks:
	All monitoring tasks (monitored components) with their defined names and a color status display
	■ Green: component works under conditions defined as safe
	■ Yellow: component works under warning zone conditions
	■ Red: Overload condition

Bar graph:
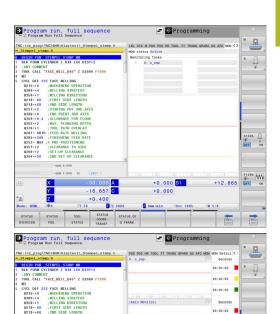
Combined display of all monitoring tasks
- ■ The red line shows the error limit defined by the machine tool builder
- ■ The yellow line shows the warning limit defined by the machine tool builder
- ■ The black line follows the condition of the component that is subjected to the highest load
  - ■ Above the red line, as soon as at least one of the monitoring tasks reaches the overload zone
  - ■ Above the green line, as soon as at least one of the monitoring tasks reaches the warning zone

Diagram zones:
- ■ Area above the red line: overload zone
- ■ Area between the red and green lines: warning zone
- ■ Area below the green line: zone of conditions defined as safe

Alternatively, the machine tool builder can define only warning limits or error limits. If no limits have been defined, the corresponding red or yellow line is not displayed.

MON Detail tab

Soft key	Meaning
No direct selection possible	Three identical areas for the display of detailed information related to three freely selectable monitoring tasks.
	The monitoring tasks can be selected via the pulldown menus above the diagrams. Once you select a component, the display indicates its name and an index (order of monitoring task definition).
	Bar graph: Individual view of the selected monitoring task ■ The red line shows the error limit defined by the machine tool builder ■ The yellow line shows the warning limit defined by the machine tool builder ■ The black line corresponds to the current load condition Alternatively, the machine tool builder can define only warning limits or error limits. If no limits have been defined, the corresponding red or yellow line is not displayed.
	Seconds: Individual display of the load duration ■ Red: duration in the overload zone ■ Yellow: duration in the warning zone ■ Green: duration in the zone defined as safe

---

i With **Component Monitoring** (option 155), the control provides automatic monitoring of configured machine components.

If configured correctly, the system will display warning messages indicating an imminent overload condition and error messages indicating that an overload condition was detected. By responding in time to these messages, you can take countermeasures to protect the machine components from damage.

If configured badly, inappropriate error messages will impede or even prevent further work. In this case, you can use the **CfgMonUser** (no. 129400) machine parameter to influence the configured overload responses.
Further information: "List of user parameter", Page 552

3.5 File management

Files

Files in the control	Type
NC programs	
in HEIDENHAIN format	.H
in DIN/ISO format	.I
Compatible NC programs	
HEIDENHAIN unit programs	.HU
HEIDENHAIN contour programs	.HC
Tables for	
Tools	.T
Tool changers	.TCH
Datums	.D
Points	.PNT
Presets	.PR
Touch probes	.TP
Backup files	.BAK
Dependent data (e.g. structure items)	.DEP
Freely definable tables	.TAB
Pallets	.P
Turning tools	.TRN
Tool compensation	.3DTC
Texts as	
ASCII files	.A
Text files	.TXT
HTML files, e.g. result logs of touch probe cycles	.HTML
Help files	.CHM
CAD files as	
ASCII files	.DXF
	.IGES
	.STEP

When you write an NC program on the control, you must first enter a program name. The control saves the NC program to the internal memory as a file with the same name. The control can also save texts and tables as files.

The control provides a special file management window in which you can easily find and manage your files. Here you can call, copy, rename and erase files.

You can manage an almost unlimited number of files with the control. The available memory is at least **21 GB**. A single NC program can be up to **2 GB** in size.

> ⓘ Depending on the setting, the control generates backup files with the extension *.bak after editing and saving of NC programs. This reduces the available memory space.

File names

When you store NC programs, tables and texts as files, the control adds an extension to the file name, separated by a point. This extension indicates the file type.

File name	File type
PROG20	.H

File names, drive names and directory names on the control must comply with the following standard: The Open Group Base Specifications Issue 6 IEEE Std 1003.1, 2004 Edition (POSIX Standard).

The following characters are permitted:

A B C D E F G H I J K L M N O P Q R S T U V W X Y Z a b c d e f g h i j k l m n o p q r s t u v w x y z 0 1 2 3 4 5 6 7 8 9 _ -

The following characters have special meanings:

Character	Meaning
.	The last period (dot) in a file name is the extension separator
\ and **/**	Directory separators
:	Separates the drive name from the directory

Do not use any other characters. This helps to prevent file transfer problems, etc.

> **ℹ** The names of tables and table columns must start with a letter and must not contain an arithmetic operator (e.g., **+**). Due to SQL commands, these characters can cause problems when inputing data or reading it out.

> **ℹ** The maximum permitted path length is 255 characters. The path length consists of the drive characters, the directory name and the file name, including the extension.
>
> **Further information:** "Paths", Page 83

Displaying externally generated files on the control

The control features several additional tools which you can use to display the files shown in the table below. Some of the files can also be edited.

File types	Type
PDF files	pdf
Excel tables	xls
	csv
Internet files	html
Text files	txt
	ini
Graphics files	bmp
	gif
	jpg
	png

Further information: "Additional tools for management of external file types", Page 94

Directories

To ensure that you can easily find your NC programs and files, we recommend that you organize your internal memory into directories (folders). You can divide a directory into further directories, which are called subdirectories. With the **-/+** key or **ENT** you can show or hide the subdirectories.

Paths

A path indicates the drive and all directories and subdirectories under which a file is saved. The individual names are separated by a backslash ****.
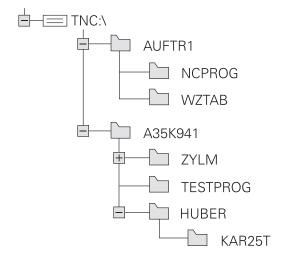
> **i** The maximum permitted path length is 255 characters. The path length consists of the drive characters, the directory name and the file name, including the extension.

Example

The directory AUFTR1 was created on the **TNC** drive. Then, in the AUFTR1 directory, the directory NCPROG was created and the NC program PROG1.H was copied into it. The NC program now has the following path:

TNC:\AUFTR1\NCPROG\PROG1.H

The chart at right illustrates an example of a directory display with different paths.

```
TNC:\
 ├─ AUFTR1
 │   ├─ NCPROG
 │   └─ WZTAB
 └─ A35K941
     ├─ ZYLM
     ├─ TESTPROG
     └─ HUBER
         └─ KAR25T
```

Calling the file manager

▶ Press the **PGM MGT** key

> The control displays the file management window (see figure for default setting. If the control displays a different screen layout, press the **WINDOW** soft key).

The narrow window on the left shows the available drives and directories. Drives designate devices with which data are stored or transferred. A drive is the internal memory of the control. Other drives are the interfaces (RS232, Ethernet) to which you can connect a PC for example. A directory is always identified by a folder symbol to the left and the directory name to the right. Subdirectories are shown to the right of and below their parent directories. If there are subdirectories, you can show or hide them using the **-/+** key.
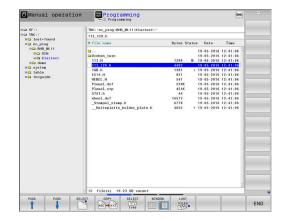
If the directory tree is longer than the screen, navigate using the scroll bar or a connected mouse.

The wide window on the right shows you all files that are stored in the selected directory. Each file is shown with additional information, illustrated in the table below.

Display	Meaning
File name	File name and file type
Bytes	File size in bytes
Status	File properties:
E	File is selected in the **Programming** operating mode
S	File is selected in the **Test Run** operating mode
M	The file is selected in a Program Run operating mode
+	File has non-displayed dependent files with the extension DEP, e.g. with use of the tool usage test
🔒	File is protected against erasing and editing
🔒	File is protected against erasing and editing, because it is being run
Date	Date that the file was last edited
Time	Time that the file was last edited

> ℹ️ To display the dependent files, set the machine parameter **dependentFiles** (no. 122101) to **MANUAL**.

Additional functions

Protecting a file and canceling file protection

▶ Place the cursor on the file you want to protect

MORE FUNCTIONS	▶ Select the additional functions: Press the **MORE FUNCTIONS** soft key
PROTECT	▶ Activate file protection: Press the **PROTECT** soft key
	▷ The file is tagged with the "protected" symbol.
UNPROTECT	▶ Cancel file protection: Press the **UNPROTECT** soft key

Selecting the editor

▶ Place the cursor on the file you want to open

MORE FUNCTIONS	▶ Select the additional functions: Press the **MORE FUNCTIONS** soft key
SELECT EDITOR	▶ Select the editor: Press the **SELECT EDITOR** soft key

▶ Mark the desired editor

- ▪ **TEXT EDITOR** for text files, e.g. **.A** or **.TXT**
- ▪ **PROGRAM EDITOR** for NC programs **.H** and **.I**
- ▪ **TABLE EDITOR** for tables, e.g. **.TAB** or **.T**
- ▪ **BPM EDITOR** for pallet tables **.P**

▶ Press the **OK** soft key

Connecting and removing USB storage devices

The control automatically detects connected USB devices with a supported file system.

To remove a USB device, proceed as follows:

MORE FUNCTIONS	▶ Move the cursor to the left-hand window
	▶ Press the **MORE FUNCTIONS** soft key
	▶ Remove the USB device

Further information: "USB devices on the control", Page 88

ADVANCED ACCESS RIGHTS

The Advanced Access Rights function can only be used in connection with user administration. This function requires the **public** directory.

Further information: "Setting advanced file access rights", Page 522

Upon the first activation of user administration, the **public** below the TNC partition will be connected.

> ℹ️   Access rights can only be defined for files located in the **public** directory.
>
> The control automatically assigns the **user** function user as the owner to any files stored on the TNC partition, but not in the **public** directory.

Further information: "'public' directory", Page 522

Selecting drives, directories and files

| PGM MGT | ▶ Call the file manager by pressing the **PGM MGT** key |

Navigate with a connected mouse or use the arrow keys or the soft keys to move the cursor to the desired position on the screen:

| ← | |
| → | ▶ Moves the cursor from the left to the right window, and vice versa |

| ↓ | ▶ Moves the cursor up and down within a window |
| ↑ | |

| PAGE ↑ | |
| PAGE ↓ | ▶ Moves the cursor one page up or down within a window |

Step 1: Select drive

▶ Move the highlight to the desired drive in the left window

| SELECT | ▶ Select a drive: Press the **SELECT** soft key, or |

| ENT | ▶ Press the **ENT** key |

Step 2: Select a directory

▶ Move the highlight to the desired directory in the left window
> The right-hand window automatically shows all files stored in the highlighted directory

Step 3: Select a file

▶ Press the **SELECT TYPE** soft key

▶ Press the **SHOW ALL** soft key
▶ Move the highlight to the desired file in the right window

▶ Press the **SELECT** soft key, or

▶ Press the **ENT** key
> The control opens the selected file in the operating mode from which you called the file manager.

ⓘ If you enter the first letter of the file you are looking for in the file manager, the cursor automatically jumps to the first NC program with the same letter.

Filtering the display

To filter the displayed files, proceed as follows:

▶ Press the **SELECT TYPE** soft key

▶ Press the soft key for the desired file type

Alternative:

▶ Press the **SHOW ALL** soft key
> The control displays all files in this folder.

Alternative:

▶ Use wildcards, such as **4*.H**
> The control will show all files of file type .h whose name starts with 4.

Alternative:

▶ Enter the file name extension, e.g. ***.H;*.D**
> The control will show all files of file type .h and .d.

Any display filter you have set will remain effective even after a control restart,

Choosing one of the last files selected

[PGM MGT] ▶ To call the file manager, press the **PGM MGT** key.

[LAST FILES] ▶ Display the last ten files selected: Press the **LAST FILES** soft key

Press the arrow keys to move the cursor to the file you wish to select:

[↑↓] ▶ Moves the cursor up and down within a window

[↕]

[OK] ▶ Select the file: Press the **OK** soft key, or

[ENT] ▶ Press the **ENT** key

> ℹ The **COPY FIELD** soft key allows you to copy the path of a marked file. You can reuse the copied path later, e.g. when calling a program with the **PGM CALL** key.

USB devices on the control
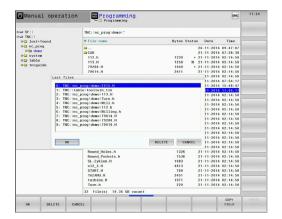
> ℹ Use the USB port only for file transfer and backup. Before editing or running an NC program, save it to the hard disk of the control. This helps to avoid duplicate data maintenance and prevents potential problems resulting from data transfer during program run.

Backing up data from or loading onto the control is especially easy with USB devices. The control supports the following USB block devices:

- Floppy disk drives with FAT/VFAT file system
- Memory sticks with the FAT/VFAT or exFAT file system
- Hard disks with the FAT/VFAT file system
- CD-ROM drives with the Joliet (ISO 9660) file system

The control automatically detects these types of USB devices when connected. The control does not support USB devices with other file systems (such as NTFS). The control displays the **USB: TNC does not support device** error message when such a device is connected.

	If an error message is displayed when connecting a USB data medium, check the setting in the SELinux security software.
	Further information: "SELinux security software", Page 494
	If the control displays the **USB: TNC does not support device** error message when using a USB hub, ignore and acknowledge the message with the **CE** key.
	If the control repeatedly fails to correctly detect a USB device with the FAT/VFAT or exFAT file system, connect another device to check the port. If this resolves the problem, use the properly working device.

Working with USB devices

	Refer to your machine manual!
	Your machine tool builder can assign permanent names for USB devices.

The USB devices appear as separate drives in the directory tree, so you can use the file management functions described in the earlier chapters.

If a larger file is transferred to a USB device in the file manager, the control displays a dialog **Write access on USB device** until file transfer is completed. The dialog is closed with the **HIDE** soft key and file transfer is continued in the background. The control displays a warning until file transfer is completed.

Removing USB devices

▶ To remove a USB device, proceed as follows:

MORE FUNCTIONS	▶ Move the cursor to the left-hand window
	▶ Press the **MORE FUNCTIONS** soft key
	▶ Remove the USB device

Data transfer to or from an external data carrier

	Before you can transfer data to an external data medium, you must set up the data interface. **Further information:** "Setting up data interfaces", Page 479

PGM MGT	▶ Press the **PGM MGT** key

WINDOW	▶ Press the **WINDOW** soft key to select the screen layout for file transfer

↕	▶ Use the arrow keys to move the cursor to the file you want to transfer
↓	> The control moves the cursor up and down within a window.
←	> The control moves the cursor from the right to the left window, and vice versa.
→	

If you wish to copy a file from the control to the external data carrier, move the cursor in the left window to the file to be transferred.

If you wish to copy from the external data medium to the control, move the cursor in the right window to the file to be transferred.

SHOW TREE	▶ Press the **SHOW TREE** soft key to select another drive or directory
	▶ Use the arrow keys to select the desired directory
SHOW FILES	▶ Press the **SHOW FILES** soft key
	▶ Use the arrow keys to select the desired file
COPY ABC→XYZ	▶ Press the **COPY** soft key
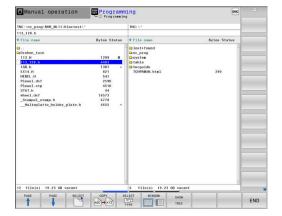
ENT	▶ Confirm with the **ENT** key
	> A status window appears on the control, informing about the copying progress.
WINDOW	▶ Alternative: Press the **WINDOW** soft key
	> The control displays the standard file manager window again.

Preventing incomplete NC programs

The control checks all NC programs for completeness before machining. The control displays a warning if the **END PGM** NC block is missing.

When you start the incomplete NC program in **Program run, single block** or **Program run, full sequence** operating mode, the control aborts with an error message.

Proceed as follows to edit the NC program:

▶ Select the NC program in **Programming** operating mode
> The control opens the NC program and automatically adds the **END PGM** NC block.
▶ Check the NC program, and complete it if necessary

SAVE AS	▶ Press the **SAVE AS** soft key
	> The control saves the NC program with the **END PGM** NC block that has been added.

The control in a network

> **i** Protect your data and your control by running your machines in a secure network.
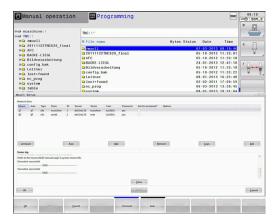
> **i** Use the Ethernet card to connect the control to the network.
> **Further information:** "Ethernet interface ", Page 485
> The control logs any error messages that occur during network operation.

If the control is connected to a network, the left directory window displays additional drives. All the functions described above (selecting a drive, copying files, etc.) also apply to network drives, provided that you have been granted the corresponding rights.

> **i** The control can also run NC programs directly from a network drive. The external drive does not provide write protection, however. This may lead to problems caused by data transfer or by a change to the NC program during machining.

Connecting and disconnecting a network drive

PGM
MGT

▶ Press the **PGM MGT** key

NET

▶ Press the **NET** soft key

▶ Press the **DEFINE NETWORK CONNECTN.** soft key.
> In a window the control shows the network drives available for access.
▶ Use the soft keys described below to define the connections for each drive

Soft key	Function
Connect	Establish the network connection. If the connection is active, the control marks the **Mount** column.
Separate	End network connection
Auto	Automatically establish network connection whenever the control is switched on. The control marks the **Auto** column if the connection is established automatically
Add	Set up new network connection
Remove	Delete existing network connection
Copy	Copy network connection
Edit	Edit network connection
Clear	Delete the status window

Data backup

HEIDENHAIN recommends backing up new NC programs and files created on the control to a PC at regular intervals.

The free-of-charge **TNCremo** software from HEIDENHAIN is a simple and convenient method for backing up data stored on the control.

You can also backup files directly from the control.
Further information: "Backup and restore", Page 473

You additionally need a data medium on which all machine-specific data, such as the PLC program, machine parameters, etc., are stored. Ask your machine manufacturer for assistance, if necessary.

> **ⓘ** Saving the contents of the internal memory can take up to several hours. In this case, it is a good idea to save the data outside of work hours, e.g. during the night.
>
> Take the time occasionally to delete any unneeded files so that the control always has enough hard-disk space for system files (such as the tool table).

> **ⓘ** Depending on operating conditions (e.g. vibration load), hard disks generally have a higher failure rate after three to five years of service. HEIDENHAIN therefore recommends having the hard disk inspected after three to five years.

Importing a file from an iTNC 530

> **⚙** Refer to your machine manual!
>
> The machine tool builder can adapt the **ADAPT NC PGM / TABLE** function.
>
> The machine tool builder can define update rules that make it possible, for example, to automatically remove umlauts from tables and NC programs.

If you export a file from an iTNC 530 and import it into a TNC 640, depending on the file type you have to adapt its format and content before you can use the file.

The machine tool builder defines which file types you will be able to import when using the **ADAPT NC PGM / TABLE** function. The control converts the contents of the imported file to a format valid for the TNC 640 and saves the changes to the selected file.

Further information: "Importing tool tables", Page 142

Additional tools for management of external file types

The additional tools enable you to display or edit various externally created file types on the control.

File types	Description
PDF files (pdf)	Page 95
Excel spreadsheets (xls, csv)	Page 96
Internet files (htm, html)	Page 97
ZIP archives (zip)	Page 99
Text files (ASCII files, e.g. txt, ini)	Page 100
Video files (ogg, oga, ogv, ogx)	Page 101
Graphic files (bmp, gif, jpg, png)	Page 101

🛈   Files with the extensions pdf, xls, zip, bmp, gif, jpg and png must be transmitted in binary format from the PC to the control. Adjust the setting in the **TNCremo** software, if required (menu item >**Extras > Configuration > Mode**).

🛈   If you are using a TNC 640 with touch control, you can replace some keystrokes with gestures.
Further information: "Operating the touchscreen", Page 533

Displaying PDF files

To open PDF files directly on the control, proceed as follows:

| PGM MGT | ▶ | Call the file manager: Press the **PGM MGT** key |

▶ Select the directory in which the PDF file is saved

▶ Move the cursor to the PDF file

| ENT | ▶ | Press the **ENT** key |
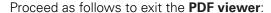
> The control opens the PDF file in its own application using the **PDF viewer** additional tool.

ⓘ With the key combination ALT+TAB, you can always return to the control's user interface while leaving the PDF file open. Alternatively, you can also click the corresponding icon in the task bar to switch back to the user interface of the control.

ⓘ If you position the mouse pointer over a button, a brief tool tip explaining the function of this button will be displayed. More information on how to use the **PDF viewer** is provided in **Help**.

Proceed as follows to exit the **PDF viewer**:

▶ Use the mouse to select the **File** menu item

▶ Select **Close**

> The control returns to the file manager.

If you are not using a mouse, proceed as follows to close the **PDF viewer**:

| ▷ | ▶ | Press the key for switching the soft keys |

> The **PDF viewer** opens the **File** pull-down menu.

| ↓ | ▶ | Move the cursor to the **Close** menu item |

| ENT | ▶ | Press the **ENT** key |

> The control returns to the file manager.

Displaying and editing Excel files

Proceed as follows to open and edit Excel files with the extension
xls, **xlsx** or **csv** directly on the control:

PGM
MGT

▶ Call the file manager: Press the **PGM MGT** key

▶ Select the directory in which the Excel file is
saved

▶ Move the cursor to the Excel file

ENT

▶ Press the **ENT** key

> The control opens the Excel file in its own
application using the **Gnumeric** additional tool.

🛈 With the key combination ALT+TAB, you can always
return to the control's user interface while leaving the
Excel file open. Alternatively, you can also click the
corresponding icon in the task bar to switch back to the
user interface of the control.

🛈 If you position the mouse pointer over a button, a brief
tool tip explaining the function of this button will be
displayed. More information on how to use **Gnumeric** is
provided in **Help**.

Proceed as follows to exit **Gnumeric**:

▶ Use the mouse to select the **File** menu item

▶ Select **Close**

> The control returns to the file manager.

If you are not using a mouse, proceed as follows to close the
additional **Gnumeric** tool:

▷

▶ Press the key for switching the soft keys

> The **Gnumeric** additional tool opens the **File** pull-
down menu.

↓

▶ Move the cursor to the **Close** menu item

ENT

▶ Press the **ENT** key

> The control returns to the file manager.

Displaying Internet files

ⓘ	The network must ensure protection against viruses and malicious software. The same applies to access to the Internet or other networks. The protective measures for this network are the responsibility of the machine tool builder and the respective network administrator, for example through a firewall.

ⓘ	Configure and use the sandbox on your control. For safety and security reasons, always open the browser in the sandbox. **Further information:** "Sandbox tab", Page 491

Proceed as follows to open Internet files with the extension **htm** or **html** directly on the control:

PGM MGT
▶ Call the file manager: Press the **PGM MGT** key
▶ Select the directory in which the Internet file is saved
▶ Move the cursor to the Internet file

ENT
▶ Press the **ENT** key
> The control opens the Internet file in its own application using the **Web Browser** additional tool.

ⓘ	With the key combination ALT+TAB, you can always return to the control's user interface while leaving the browser open. Alternatively, you can also click the corresponding icon in the task bar to switch back to the user interface of the control.

ⓘ	If you position the mouse pointer over a button, a brief tool tip explaining the function of this button will be displayed. More information on how to use **Web Browser** is available in **Help**.

When you start the web browser, it checks at regular intervals whether updates are available.

You can update the web browser only if you deactivate the SELinux security software during this time and you are connected to the Internet.

ⓘ	Reactivate SELinux after running the update.

Proceed as follows to close the **web browser**:

▶ Use the mouse to select the **File** menu item

▶ Select **Quit**

> The control returns to the file manager.

If you are not using a mouse, proceed as follows to close the **web browser**:

▷	▶ Press the key for switching the soft keys: The **web browser** opens the **File** pull-down menu
↓	▶ Move the cursor to the **Quit** menu item
ENT	▶ Press the **ENT** key > The control returns to the file manager.

Working with ZIP archives

Proceed as follows to open ZIP archives with the extension **zip** directly on the control:

PGM MGT	▶	Call the file manager: Press the **PGM MGT** key
	▶	Select the directory in which the archive file is saved
	▶	Move the cursor to the archive file
ENT	▶	Press the **ENT** key
	>	The control opens the archive file in its own application using the **Xarchiver** additional tool.
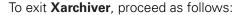
> ⓘ With the key combination ALT+TAB, you can always return to the control's user interface while leaving the archive file open. Alternatively, you can also click the corresponding icon in the task bar to switch back to the user interface of the control.

> ⓘ If you position the mouse pointer over a button, a brief tool tip explaining the function of this button will be displayed. More information on how to use the **Xarchiver** function is provided under **Help**.

To exit **Xarchiver**, proceed as follows:
- ▶ Use the mouse to select the **ARCHIVE** menu item
- ▶ Select **Exit**
- > The control returns to the file manager.

If you are not using a mouse, proceed as follows to close the **Xarchiver**:
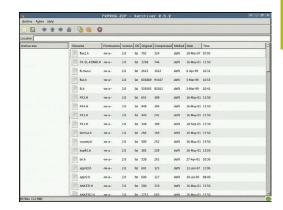
▷	▶	Press the key for switching the soft keys
	>	**Xarchiver** opens the **ARCHIVE** pull-down menu.
↓	▶	Move the cursor to the **Exit** menu item**Exit**
ENT	▶	Press the **ENT** key
	>	The control returns to the file manager.

Displaying or editing text files

Use the internal text editor to open and edit text files (ASCII files, e.g. with the extension **txt**). Proceed as follows:

PGM
MGT

▶ Call the file manager: Press the **PGM MGT** key
▶ Select the drive and the directory in which the text file is saved
▶ Move the cursor to the text file

ENT

▶ Press the **ENT** key
> The control opens the text file with the internal text editor.

ℹ Alternatively, you can also open the ASCII files using the **Leafpad** additional tool. The shortcuts you are familiar with from Windows, which you can use to edit texts quickly (CTRL+C, CTRL+V,...), are available within **Leafpad**.

ℹ With the key combination ALT+TAB, you can always return to the control's user interface while leaving the text file open. Alternatively, you can also click the corresponding icon in the task bar to switch back to the user interface of the control.

Proceed as follows to open **Leafpad**:

▶ Use the mouse to select the **Menu** HEIDENHAIN icon from the task bar
▶ Select the **Tools** and **Leafpad** menu items in the pull-down menu

Proceed as follows to exit **Leafpad**:

▶ Use the mouse to select the **File** menu item
▶ Select **Exit**
> The control returns to the file manager.

Displaying video files

> ⚙ This function must be enabled and adapted by the machine tool builder.

Proceed as follows to open video files with the extension **ogg**, **oga**, **ogv** or **ogx** directly on the control:

PGM MGT	▶ Call the file manager: Press the **PGM MGT** key
	▶ Select the directory in which the video file is saved
	▶ Move the cursor to the video file
ENT	▶ Press the **ENT** key
	> The control opens the video file in its own application.

> ℹ The paid-for Fluendo Codec Pack is required for other formats, such as for MP4 files.

> ⚙ Any additional software will be installed by your machine tool builder.

Displaying graphics files

Proceed as follows to open graphics files with the extension **bmp**, **gif**, **jpg** or **png** directly on the control:

PGM MGT	▶ Call the file manager: Press the **PGM MGT** key
	▶ Select the directory in which the graphics file is saved
	▶ Move the cursor to the graphics file
ENT	▶ Press the **ENT** key
	> The control opens the graphics file in its own application using the **Ristretto** additional tool.

> ℹ With the key combination ALT+TAB, you can always return to the control's user interface while leaving the graphics file open. Alternatively, you can also click the corresponding icon in the task bar to switch back to the user interface of the control.

> ℹ More information on how to use the **ristretto** function is provided under **Help**.

Proceed as follows to exit **Ristretto**:

▶ Use the mouse to select the **File** menu item
▶ Select **Exit**
> The control returns to the file manager.

If you are not using a mouse, proceed as follows to close the
Ristretto tool:

▶ Press the key for switching the soft keys
> **Ristretto** opens the **File** pull-down menu.
▶ Move the cursor to the **Exit** menu item

▶ Press the **ENT** key
> The control returns to the file manager.

3.6 Error messages and help system

Error messages

Display of errors

The control displays error messages in the following cases, for example:

- Incorrect data input
- Logical errors in the NC program
- Contour elements that are impossible to machine
- Incorrect use of touch probes

When an error occurs, the control displays it in red type in the header.

> **i** The control uses different colors for different error classes:
>
> - red for errors
> - yellow for warnings
> - green for notes
> - blue for information

Long and multi-line error messages are displayed in abbreviated form. Complete information on all pending errors is shown in the error window.

The control displays an error message in the header until it is cleared or replaced by a higher-priority error (higher error class). Information that appears only briefly is always displayed.

An error message that contains an NC block number was caused by an error in the indicated NC block or in the preceding NC block.

If a rare **processor check error** should occur, the control automatically opens the error window. You cannot correct such an error. Shut down the system and restart the control.

Opening the error window

`ERR`
- ▶ Press the **ERR** key
- > The control opens the error window and displays all accumulated error messages.

Closing the error window

`END`
- ▶ Press the **END** soft key**END**

`ERR`
- ▶ Alternative: Press the **ERR** key
- > The control closes the error window.
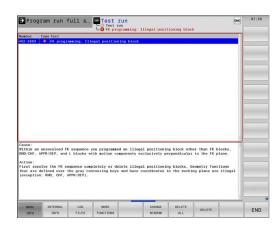
Detailed error messages

The control displays possible causes of the error and suggestions for solving the problem:

▶ Open the error window

MORE
INFO

▶ Information on the error cause and corrective action: Position the cursor on the error message and press the **MORE INFO** soft key

\> The control opens a window with information on the error cause and corrective action.

▶ Leave Info: Press the **MORE INFO** soft key again

Soft key: INTERNAL INFO

The **INTERNAL INFO** soft key supplies information on the error message. This information is only required if servicing is needed.

▶ Open the error window

INTERNAL
INFO

▶ Detailed information about the error message: Position the cursor on the error message and press the **INTERNAL INFO** soft key

\> The control opens a window with internal information about the error.

▶ Exit the details: Press the **INTERNAL INFO** soft key again

Soft key FILTER

The **FILTER** soft key enables you to filter identical warnings listed immediately in succession.

▶ Open the error window

MORE
FUNCTIONS

▶ Press the **MORE FUNCTIONS** soft key

FILTER
OFF   ON

▶ Press the **FILTER** soft key

\> The control filters the identical warnings.

▶ Exit the filter: Press the **GO BACK** soft key

ACTIVATE AUTOMATIC SAVING soft key

Using the **ACTIVATE AUTOMATIC SAVING** soft key, you can specify error numbers that cause the control to save a service file if an error with that number occurs.

▶ Open the error window

| MORE FUNCTIONS | ▶ Press the **MORE FUNCTIONS** soft key |

| ACTIVATE AUTOMATIC SAVING | ▶ Press the **ACTIVATE AUTOMATIC SAVING** soft key
> The control opens the **ACTIVATE AUTOMATIC SAVING** pop-up window.
▶ Define the entries
 ■ **Error number**: Enter the desired error number
 ■ **active:** Enable this option to automatically create the service file
 ■ **Comment:** Enter a comment on this error number, if required |

| STORE | ▶ Press the **STORE** soft key
> If an error with the specified error number occurs, a service file will be saved automatically. |

| | ▶ Press the **GO BACK** soft key |

Clearing errors

Clearing errors automatically

> The control will automatically clear pending warning or error messages when a new NC program is selected or the previous one is restarted. The machine tool builder specifies in the optional machine parameter **CfgClearError** (no. 130200) whether these messages will be cleared automatically or not.
>
> The factory default setting of the control defines that warning and error messages in the **Test Run** and **Programming** operating modes will be cleared automatically from the error window. Messages issued in the machine operating modes will not be cleared.

Clearing errors outside of the error window

| CE | ▶ Clear the errors/messages in the header: Press the **CE** key |

> In certain situations you cannot use the **CE** key for clearing the errors because the key is used for other functions.

Clearing errors

▶ Open the error window

| DELETE | ▶ Clear individual error messages: Position the cursor on the error message and press the **DELETE** soft key. |

| DELETE ALL | ▶ Clear all error messages: Press the **DELETE ALL** soft key. |

| **ⓘ** | If the cause of the error has not been corrected, the error message cannot be cleared. In this case, the error message remains in the window. |

Error log

The control stores errors occurred and important events (e.g. system start) in an error log. The capacity of the error log is limited. If the log is full, the control uses a second file. If this is also full, the first error log is deleted and newly written etc. If required, switch from **CURRENT FILE** to **PREVIOUS FILE** to view the history.

▶ Open the error window.

| LOG FILES | ▶ Press the **LOG FILES** soft key |

| ERROR LOG | ▶ Open the error log file: Press the **ERROR LOG** soft key |

| PREVIOUS FILE | ▶ Set the previous error log if required: Press the **PREVIOUS FILE** soft key |

| CURRENT FILE | ▶ Set the current error log if required: Press the **CURRENT FILE** soft key |

The oldest entry is at the beginning of the log file, and the most recent entry is at the end.

Keystroke log

The control stores each key pressed and important events (e.g. system start) in a keystroke log. The capacity of the keystroke log is limited. If the keystroke log is full, the control switches to a second keystroke log. If this is also full, the first keystroke log is deleted and newly written etc. If required, switch from **CURRENT FILE** to **PREVIOUS FILE** to view the history of the inputs.

LOG FILES	▶ Press the **LOG FILES** soft key
KEYSTROKE LOG	▶ Open the keystroke log file: Press the **KEYSTROKE LOG** soft key
PREVIOUS FILE	▶ Set the previous keystroke log if required: Press the **PREVIOUS FILE** soft key
CURRENT FILE	▶ Set the current keystroke log if required: Press the **CURRENT FILE** soft key

The control saves each key pressed during operation in a keystroke log. The oldest entry is at the beginning, and the most recent entry is at the end of the file.

Overview of the keys and soft keys for viewing the log

Soft key/ Keys	Function
BEGIN	Go to beginning of keystroke log
END	Go to end of keystroke log
FIND	Find text
CURRENT FILE	Current keystroke log
PREVIOUS FILE	Previous keystroke log
↕ ↕	Up/down one line
	Return to main menu

Informational texts

If an operating error occurred, e.g. pressing an impermissible key or entering a value outside of a validity range, the control displays an information text in the header to inform you of the operating error. The control deletes this information text with the next valid entry.

Saving service files

If necessary, you can save the current status of the control and make it available to a service technician for evaluation. A group of service files is saved (error and keystroke logs as well as other files that contain information about the current status of the machine and the machining).

ℹ️ In order to facilitate sending service files via email, the control will only save active NC programs with a size of up to 10 MB in the service file. If the NC program is bigger, it will not be added to the created service file.

If you repeat the **SAVE SERVICE FILES** function with the same file name, the previously saved group of service files is overwritten. Therefore, use another file name when executing the function another time.

Saving service files

▶ Open the error window

| LOG FILES |
▶ Press the **LOG FILES** soft key

| SAVE SERVICE FILES |
▶ Press the **SAVE SERVICE FILES** soft key
> The control opens a pop-up window in which you can enter a file name or a complete path for the service file.

| OK |
▶ Save the service files: Press the **OK** soft key

Calling the TNCguide help system

You can call the control's help system via soft key. Immediately the help system shows you the same error explanation that you receive by pressing the **HELP** soft key.

⚙️ Refer to your machine manual!

If your machine manufacturer also provides a help system, the control shows an additional **Machine manufacturer (OEM)** soft key with which you can call this separate help system. There you will find further, more detailed information on the error message concerned.

| HEIDENHAIN TNCguide |
▶ Call the help for HEIDENHAIN error messages

| MACHINE MFR. |
▶ Call the help for HEIDENHAIN machine-specific error messages, if available

TNCguide context-sensitive help system

Application

> ℹ️ Before you can use the TNCguide, you need to download the help files from the HEIDENHAIN home page
>
> **Further information:** "Downloading current help files", Page 114

The **TNCguide** context-sensitive help system contains the user documentation in HTML format. The TNCguide is called with the **HELP** key, and the control often immediately displays the information specific to the condition from which the help was called (context-sensitive call). Even if you are editing an NC block and press the **HELP** key, you are usually brought to the exact place in the documentation that describes the corresponding function.

> ℹ️ The control tries to start the TNCguide in the language that you have selected as the conversational language. If the required language version is not available, the control automatically opens the English version.
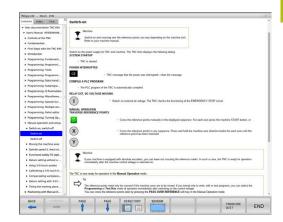
The following user documentation is available in TNCguide:

- Conversational Programming User's Manual (**BHBKlartext.chm**)
- ISO User's Manual (**BHBIso.chm**)
- User's Manual for Setup, Testing and Running NC Programs (**BHBoperate.chm**)
- User's Manual for Cycle Programming (**BHBtchprobe.chm**)
- List of All Error Messages **(errors.chm)**

In addition, the **main.chm** "book" file is available, in which all existing .chm files are shown in one place.

> ⚙️ As an option, your machine tool builder can embed machine-specific documentation in the **TNCguide**. These documents then appear as a separate book in the **main.chm** file.

Working with TNCguide

Calling TNCguide

There are several ways to start the TNCguide:

▶ Press the **HELP** key.

▶ Click the help symbol at the lower right of the screen beforehand, then click the appropriate soft keys

▶ Open a help file (CHM file) via the file management. The control can open any .chm file, even if it is not saved in the control's internal memory
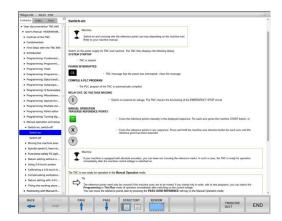
> ⓘ On the Windows programming station, the TNCguide is opened in the internally defined standard browser.

For many soft keys there is a context-sensitive call through which you can go directly to the description of the soft key's function. This functionality requires using a mouse. Proceed as follows:

▶ Select the soft-key row containing the desired soft key

▶ Click with the mouse on the help symbol that the control displays just above the soft-key row

> The mouse pointer turns into a question mark.

▶ Move the question mark to the soft key for which you want an explanation

> The control opens the TNCguide. If there is no entry point for the selected soft key, then the control opens the book file **main.chm**. You can search for the desired explanation using full text search or by using the navigation.

Even if you are editing an NC block, context-sensitive help is available:

▶ Select any NC block

▶ Select the desired word

▶ Press the **HELP** key.

> The control opens the Help system and shows the description of the active function. This does not apply for miscellaneous functions or cycles from your machine manufacturer.

Navigating in the TNCguide

It's easiest to use the mouse to navigate in TNCguide. A table of contents appears on the left side of the screen. By clicking the rightward pointing triangle you open subordinate sections, and by clicking the respective entry you open the individual pages. It is operated in the same manner as the Windows Explorer.

Linked text positions (cross references) are shown underlined and in blue. Clicking the link opens the associated page.

Of course you can also operate TNCguide through keys and soft keys. The following table contains an overview of the corresponding key functions.

Soft key	Function
↑ ↓	■ If the table of contents at left is active: Select the entry above it or below it ■ If the text window at right is active: Move the page downward or upward if texts or graphics are not shown completely
→	■ If the table of contents at left is active: Open up the table of contents ■ If the text window at right is active: No function
←	■ If the table of contents at left is active: Close the table of contents ■ If the text window at right is active: No function
ENT	■ If the table of contents at left is active: Use the cursor key to show the selected page ■ If the text window at right is active: If the cursor is on a link, jump to the linked page
	■ If the table of contents at left is active: Switch the tab between the display of the table of contents, display of the subject index, and the full-text search function and switching to the screen half at right ■ If the text window at right is active: Jump back to the window at left
	■ If the table of contents at left is active: Select the entry above it or below it ■ If the text window at right is active: Jump to next link
BACK	Select the page last shown
FORWARD	Page forward if you have used the **Select page last shown** function
PAGE ↑	Move up by one page
PAGE ↓	Move down by one page

Soft key	Function
DIRECTORY	Display or hide table of contents
WINDOW	Switch between full-screen display and reduced display. With the reduced display you can see some of the rest of the control window
SWITCH	The focus is switched internally to the control application so that you can operate the control when the TNCguide is open. If the full screen is active, the control reduces the window size automatically before the change of focus
END	Exit TNCguide

Subject index

The most important subjects in the Manual are listed in the subject index (**Index** tab). You can select them directly by mouse or with the arrow keys.

The left side is active.

▶ Select the **Index** tab
▶ Use the arrow keys or the mouse to select the desired keyword

Alternative:

▶ Enter the first few characters
> The control synchronizes the subject index and creates a list in which you can find the subject more easily.
▶ Use the **ENT** key to call the information on the selected keyword

Full-text search

On the **Find** tab, you can search all of TNCguide for a specific word.

The left side is active.

▶ Select the **Find** tab

▶ Activate the **Find:** entry field

▶ Enter the search word

▶ Press the **ENT** key

> The control lists all sources containing the word.

▶ Use the arrow keys to navigate to the desired source

▶ Press the **ENT** key to go to the selected source

ⓘ The full-text search only works for single words.

If you activate the **Search only in titles** function, the control searches only through headings and ignores the body text. To activate the function, use the mouse or select it and then press the space bar to confirm.

Downloading current help files

You'll find the help files for your control software on the HEIDENHAIN homepage:
http://content.heidenhain.de/doku/tnc_guide/html/en/ index.html

Navigate to the suitable help file as follows:

▶ TNC Controls

▶ Series, e.g. TNC 600

▶ Desired NC software number, e.g.TNC 640 (34059x-10)

▶ Select the desired language version from the **TNCguide online help** table

▶ Download the ZIP file

▶ Extract the ZIP file

▶ Move the extracted CHM files to the **TNC:\tncguide\en** directory or the respective language subdirectory on the control

---

> ℹ️ When using **TNCremo** to transfer the CHM files to the control, select the binary mode for files with the **.chm** extension.

Language	TNC directory
German	**TNC:\tncguide\de**
English	**TNC:\tncguide\en**
Czech	**TNC:\tncguide\cs**
French	**TNC:\tncguide\fr**
Italian	**TNC:\tncguide\it**
Spanish	**TNC:\tncguide\es**
Portuguese	**TNC:\tncguide\pt**
Swedish	**TNC:\tncguide\sv**
Danish	**TNC:\tncguide\da**
Finnish	**TNC:\tncguide\fi**
Dutch	**TNC:\tncguide\nl**
Polish	**TNC:\tncguide\pl**
Hungarian	**TNC:\tncguide\hu**
Russian	**TNC:\tncguide\ru**
Chinese (simplified)	**TNC:\tncguide\zh**
Chinese (traditional)	**TNC:\tncguide\zh-tw**
Slovenian	**TNC:\tncguide\sl**
Norwegian	**TNC:\tncguide\no**
Slovak	**TNC:\tncguide\sk**
Korean	**TNC:\tncguide\kr**
Turkish	**TNC:\tncguide\tr**
Romanian	**TNC:\tncguide\ro**

3.7 NC fundamentals
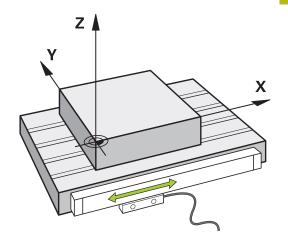
Position encoders and reference marks
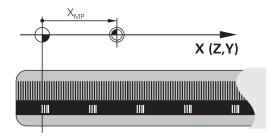
The machine axes are equipped with position encoders that register the positions of the machine table or tool. Linear axes are usually equipped with linear encoders, rotary tables and tilting axes with angle encoders.

When a machine axis moves, the corresponding position encoder generates an electrical signal. The control evaluates this signal and calculates the precise actual position of the machine axis.

If there is a power interruption, the calculated position will no longer correspond to the actual position of the machine slide. To recover this assignment, incremental position encoders are provided with reference marks. When a reference mark is crossed over, a signal identifying a machine-based reference point is transmitted to the control. This enables the control to re-establish the assignment of the displayed position to the current machine position. For linear encoders with distance-coded reference marks, the machine axes need to move by no more than 20 mm, for angle encoders by no more than 20°.

With absolute encoders, an absolute position value is transmitted to the control immediately upon switch-on. In this way the assignment of the actual position to the machine slide position is re-established directly after switch-on.

Programmable axes

In the default setting, the programmable axes of the control are in accordance with the axis definitions specified in DIN 66217.

The designations of the programmable axes are given in the table below.
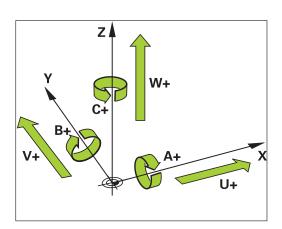
Principal axis	Parallel axis	Rotary axis
X	U	A
Y	V	B
Z	W	C

⚙ Refer to your machine manual!

The number, designation and assignment of the programmable axes depend on the machine.

Your machine tool builder can define further axes, such as PLC axes.

Reference systems

For the control to move an axis in accordance with a defined path, it requires a **reference system**.

A paraxially mounted linear encoder on a machine tool serves as a simple reference system for linear axes. The linear encoder represents a **number ray**, a one-dimensional coordinate system.

To approach a point on the **plane**, the control requires two axes and therefore a reference system with two dimensions.

To approach a point in **space**, the control requires three axes and therefore a reference system with three dimensions. If these three axes are arranged perpendicularly to each other, this creates a **three-dimensional Cartesian coordinate system**.

> **i**   According to the right-hand rule the fingertips point in the positive directions of the three principal axes.

For a point to be uniquely determined in space, a **coordinate origin** is needed in addition to the arrangement of the three dimensions. The common intersection serves as the coordinate origin in a 3-D coordinate system. This intersection has the coordinates **X+0**, **Y+0**, and **Z+0**.

In order, for example, for the control to always perform a tool change at the same position, as well as always execute a machining operation referenced to the current workpiece position, the control must be able to differentiate between different reference systems.
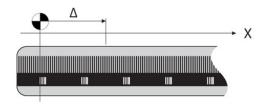
The control differentiates between the following reference systems:
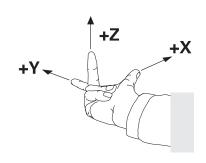
- Machine coordinate system M-CS:
  Machine **C**oordinate **S**ystem
- Basic coordinate system B-CS:
  Basic **C**oordinate **S**ystem
- Workpiece coordinate system W-CS:
  Workpiece **C**oordinate **S**ystem
- Working plane coordinate system WPL-CS:
  Working **Pl**ane **C**oordinate **S**ystem
- Input coordinate system I-CS:
  Input **C**oordinate **S**ystem
- Tool coordinate system T-CS:
  Tool **C**oordinate **S**ystem
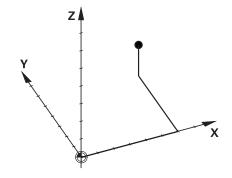
> **i**   All reference systems build upon each other. They are subject to the kinematic chain of the specific machine tool.
>
> The machine coordinate system is the reference system.

Machine coordinate system M-CS
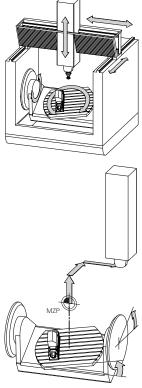
The machine coordinate system corresponds to the description of kinematics and therefore to the actual mechanical design of the machine tool.

Because the mechanics of a machine tool never precisely correspond to a Cartesian coordinate system, the machine coordinate system consists of several one-dimensional coordinate systems. These one-dimensional coordinate systems correspond to the physical machine axes that are not necessarily perpendicular to each other.

The position and orientation of the one-dimensional coordinate systems are defined with the aid of translations and rotations based on the spindle tip in the description of kinematics.

The position of the coordinate origin, the machine datum, is defined by the machine manufacturer during machine configuration. The values in the machine configuration define the zero positions of the encoders and the corresponding machine axes. The machine datum does not necessarily have to be located in the theoretical intersection of the physical axes. It can therefore also be located outside of the traverse range.

Because the machine configuration values cannot be modified by the user, the machine coordinate system is used for determining constant positions, e.g. the tool change point.

Machine datum (MZP)

Soft key	Application
BASE TRANSFORM. OFFSET	The user can define shifts in the machine coordinate system according to the specific axis with use of the **OFFSET** values of the preset table.

> ⚙ The machine tool builder configures the **OFFSET** columns of the preset management in accordance with the machine.

Further information: "Preset management", Page 194

⚠ *NOTICE*

Danger of collision!

Your control may feature an additional pallet preset table, depending on the machine. In this table the machine tool builder can define **OFFSET** values that take effect before the **OFFSET** values you specify in the preset table become effective. The **PAL** tab of the additional status display indicates whether a pallet preset is active, and which one. Since the **OFFSET** values of the pallet preset table are neither shown nor editable, there is a risk of collision during all movements!

▸ Refer to the machine tool builder's documentation
▸ Use pallet presets only in conjunction with pallets
▸ Check the display of the **PAL** tab before you start machining

ℹ   The **Global Program Settings** function (Option 44) additionally provides the **Additive offset (M-CS)** transformation for tilting axes. This transformation is added to the **OFFSET** values from the preset table and pallet preset table.

ℹ   Another feature is **OEM-OFFSET**, which is available only to the machine tool builder. **OEM-OFFSET** can be used to define additive axis shifts for rotary and parallel axes.

The sum of all **OFFSET** values (from all the above **OFFSET** input possibilities) results in the difference between the **ACTL.** position and the **RFACTL** position of an axis.

The control converts all movements in the machine coordinate system, independent of the reference system used for value input.

Example of a 3-axis machine tool with a Y axis as oblique axis, not arranged perpendicularly to the ZX plane:

▶ In the **Positioning w/ Manual Data Input** operating mode, run an NC block with **L IY+10**

> The control determines the required axis nominal values from the defined values.

> During positioning the control moves the **Y and Z** machine axes.

> The **RFACTL** and **RFNOML** displays show movements of the Y axis and Z axis in the machine coordinate system.

> The **ACTL.** and **NOML.** displays only show one movement of the Y axis in the input coordinate system.

▶ In the **Positioning w/ Manual Data Input** operating mode, run an NC block with **L IY-10 M91**

> The control determines the required axis nominal values from the defined values.

> During positioning the control only moves the **Y** machine axis.

> The **RFACTL** and **RFNOML** displays only show one movement of the Y axis in the machine coordinate system.

> The **ACTL.** and **NOML.** displays show movements of the Y axis and Z axis in the input coordinate system.

The user can program positions related to the machine datum, e.g. by using the miscellaneous function **M91**.

Basic coordinate system B-CS

The basic coordinate system is a 3-D Cartesian coordinate system. Its coordinate origin is the end of the kinematics model.

The orientation of the basic coordinate system in most cases corresponds to that of the machine coordinate system. There may be exceptions to this if a machine manufacturer uses additional kinematic transformations.

The kinematic model and thus the position of the coordinate origin for the basic coordinate system is defined by the machine manufacturer in the machine configuration. The user cannot modify the machine configuration values.

The basic coordinate system serves to determine the position and orientation of the workpiece coordinate system.

Soft key	Application
BASE TRANSFORM. OFFSET	The user determines the position and orientation of the workpiece coordinate system by using a 3-D touch probe for example. The control saves the values determined with respect to the basic coordinate system as **BASE TRANSFORM.** values in the preset management.

The machine tool builder configures the **BASE TRANSFORM.** columns of the preset management in accordance with the machine.
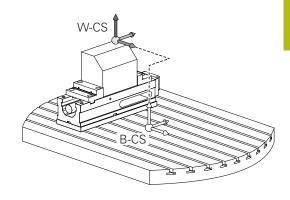
NOTICE

Danger of collision!
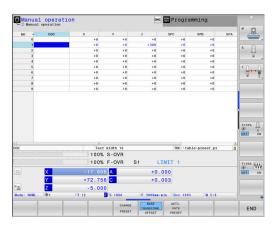
Your control may feature an additional pallet preset table, depending on the machine. In this table the machine tool builder can define **BASE TRANSFORM.** values that take effect before the **BASE TRANSFORM.** values you specify in the preset table become effective. The **PAL** tab of the additional status display indicates whether a pallet preset is active, and which one. Since the **BASE TRANSFORM.** values of the pallet preset table are neither visible nor editable, there is danger of collision during all movements!

▶ Refer to the machine tool builder's documentation
▶ Use pallet presets only in conjunction with pallets
▶ Check the display of the **PAL** tab before you start machining

Workpiece coordinate system W-CS

The workpiece coordinate system is a 3-D Cartesian coordinate system. Its coordinate origin is the active reference point.

The position and orientation of the workpiece coordinate system depend on the **BASE TRANSFORM.** values of the active line in the preset table.

Soft key	Application
BASE TRANSFORM. OFFSET	The user determines the position and orientation of the workpiece coordinate system by using a 3-D touch probe for example. The control saves the values determined with respect to the basic coordinate system as **BASE TRANSFORM.** values in the preset management.

Further information: "Preset management", Page 194

---
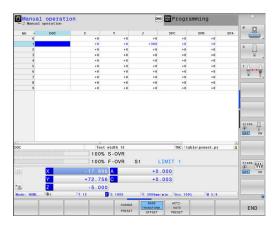
i The **Global Program Settings** function (Option 44) additionally provides the following transformations:
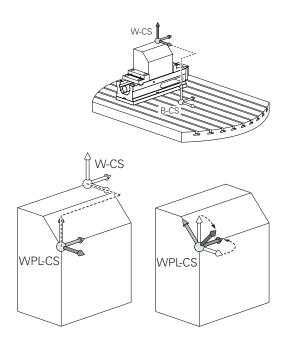
- The **Additive basic rotat. (W-CS)** is added to a basic rotation or a 3-D basic rotation from the preset table and the pallet preset table. The **Additive basic rotat. (W-CS)** is the first transformation that is possible in the workpiece coordinate system (W-CS).
- **Shift (W-CS)** is added to the shift (Cycle 7 **DATUM SHIFT**) that is defined in the NC program before tilting the working plane.
- **Mirroring** is added to the mirroring (Cycle 8 **MIRRORING**) that is defined in the NC program before tilting the working plane.
- **Shift (mW-CS)** is effective in the "modified workpiece coordinate system" after applying the **Shift (W-CS)** or **Mirroring (W-CS)** transformation and before tilting the working plane.

In the workpiece coordinate system the user defines the position and orientation of the working plane coordinate system with use of transformations.

Transformations in the workpiece coordinate system:
- **3D ROT** functions
  - **PLANE** functions
  - Cycle 19 **WORKING PLANE**
- Cycle 7 **DATUM SHIFT**
  (shifting **before** tilting the working plane)
- Cycle 8 **MIRROR IMAGE**
  (mirroring **before** tilting the working plane)

i The result of transformations built up on each other depends on the programming sequence.

In every coordinate system, program only the specified (recommended) transformations. This applies to both setting and resetting the transformations. Any other use may lead to unexpected or undesired results. Please observe the following programming notes.

Programming notes:

■ Transformations (mirroring and shifting) that are programmed before the **PLANE** functions (except for **PLANE AXIAL**) will change the position of the tilt datum (origin of the working plane coordinate system WPL-CS) and the orientation of the rotary axes

   ■ If you just program a shift, then only the position of the tilt datum will change

   ■ If you just program mirroring, then only the orientation of the rotary axes will change

■ When used in conjunction with **PLANE AXIAL** and Cycle 19, the programmed transformations (mirroring, rotation and scaling) do not affect the position of the tilt datum or the orientation of the rotary axes

i Without active transformations in the workpiece coordinate system, the position and orientation of the working plane coordinate system and workpiece coordinate system are identical.

There are no transformations in the workpiece coordinate system on 3-axis machine tools or with pure 3-axis machining. The **BASE TRANSFORM.** values of the active line of the preset table have a direct effect on the working plane coordinate system with this assumption.

Other transformations are of course possible in the working plane coordinate system

Further information: "Working plane coordinate system WPL-CS", Page 122

Working plane coordinate system WPL-CS

The working plane coordinate system is a 3-D Cartesian coordinate system.

The position and orientation of the working plane coordinate system depend on the active transformations in the workpiece coordinate system.

> **i**   Without active transformations in the workpiece coordinate system, the position and orientation of the working plane coordinate system and workpiece coordinate system are identical.
>
> There are no transformations in the workpiece coordinate system on 3-axis machine tools or with pure 3-axis machining. The **BASE TRANSFORM.** values of the active line of the preset table have a direct effect on the working plane coordinate system with this assumption.

In the working plane coordinate system the user defines the position and orientation of the input coordinate system with use of transformations.

> **i**   The **Mill-Turning** function (option 50) additionally provides the **OEM rotation** and **precession angle** transformations.
>
> ■ **OEM rotation** is available only to the machine tool builder and takes effect before the **precession angle**
>
> ■ **Precession angle** is defined in Cycles 800 **ADJUST XZ SYSTEM**, 801 **RESET ROTARY COORDINATE SYSTEM** and 880 **GEAR HOBBING**, and takes effect before the other transformations of the working plane coordinate system
>
> The active values of the two transformations (if not equal to 0) are shown on the **POS** tab of the additional status display. Check the values also in milling mode because any active transformations will also remain active in that mode!
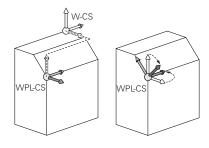
> **⚙**   Refer to your machine manual!
>
> Your machine tool builder can use the **OEM rotation** and **precession angle** transformations also without the **Mill-Turning** function (option 50).
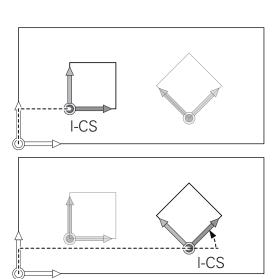
Transformations in the working plane coordinate system:

■ Cycle 7 **DATUM SHIFT**

■ Cycle 8 **MIRROR IMAGE**

■ Cycle 10 **ROTATION**

■ Cycle 11 **SCALING**

■ Cycle 26 **AXIS-SPECIFIC SCALING**

■ **PLANE RELATIVE**

ⓘ　As a **PLANE** function, the **PLANE RELATIVE** is effective in the workpiece coordinate system and aligns the working plane coordinate system.

　　The values of additive tilting always relate to the current working plane coordinate system.

ⓘ　The **Global Program Settings** function (Option 44) additionally provides the **Rotation (WPL-CS)** transformation. This transformation is added to the rotation (Cycle 10 **ROTATION**) that is defined in the NC program.

ⓘ　The result of transformations built up on each other depends on the programming sequence.

ⓘ　Without active transformations in the working plane coordinate system, the position and orientation of the input coordinate system and working plane coordinate system are identical.

　　In addition, there are no transformations in the workpiece coordinate system on 3-axis machine tools or with pure 3-axis machining. The **BASE TRANSFORM.** values of the active line of the preset table have a direct effect on the input coordinate system with this assumption.

Input coordinate system I-CS

The input coordinate system is a 3-D Cartesian coordinate system.

The position and orientation of the input coordinate system depend on the active transformations in the working plane coordinate system.

> **i** Without active transformations in the working plane coordinate system, the position and orientation of the input coordinate system and working plane coordinate system are identical.
>
> In addition, there are no transformations in the workpiece coordinate system on 3-axis machine tools or with pure 3-axis machining. The **BASE TRANSFORM.** values of the active line of the preset table have a direct effect on the input coordinate system with this assumption.

With the aid of positioning blocks in the input coordinate system, the user defines the position of the tool and therefore the position of the tool coordinate system.

> **i** The **NOML.**, **ACTL.**, **LAG** and **ACTDST** displays are also based on the input coordinate system.

Positioning blocks in input coordinate system:

- ■ Paraxial positioning blocks
- ■ Positioning blocks with Cartesian or polar coordinates
- ■ Positioning blocks with Cartesian coordinates and surface normal vectors

Example

7 X+48 R+

7 L X+48 Y+102 Z-1.5 R0

7 LN X+48 Y+102 Z-1.5 NX-0.04658107 NY0.00045007 NZ0.8848844 R0

> **i** The position of the tool coordinate system is determined by the Cartesian coordinates X, Y and Z also for positioning blocks with surface normal vectors.
>
> In conjunction with 3-D tool compensation, the position of the tool coordinate system can be shifted along the surface normal vectors.
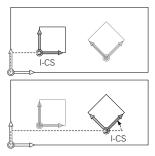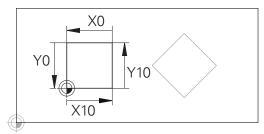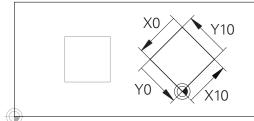
> **i** Orientation of the tool coordinate system can be performed in various reference systems.
>
> **Further information:** "Tool coordinate system T-CS", Page 125

A contour referencing the input coordinate system origin can easily be transformed any way you need.

Tool coordinate system T-CS

The tool coordinate system is a 3-D Cartesian coordinate system. Its coordinate origin is the tool reference point. The values of the tool table, **L** and **R** with milling tools and **ZL**, **XL** and **YL** with turning tools, reference this point.

Further information: "Entering tool data into the table", Page 137 and "Tool data", Page 400

> **i** For dynamic collision monitoring (option 40) to correctly monitor the tool, the values in the tool table must correspond to the actual dimensions of the tool.

In accordance with the values from the tool table, the coordinate origin of the tool coordinate system is shifted to the tool center point TCP. TCP stands for **T**ool **C**enter **P**oint.

If the NC program does not reference the tool tip, the tool center point must be shifted. The required shift is implemented in the NC program using the delta values during a tool call.

> **i** The position of the TCP as shown in the diagram is obligatory in conjunction with the 3-D tool compensation.

> **i** With the aid of positioning blocks in the input coordinate system, the user defines the position of the tool and therefore the position of the tool coordinate system.

If the **TCPM** function or miscellaneous function **M128** is active, the orientation of the tool coordinate system depends on the tool's current angle of inclination.

The user defines the tool's angle of inclination either in the machine coordinate system or in the working plane coordinate system.

Tool angle of inclination in the machine coordinate system:

Example

7	L X+10 Y+45 A+10 C+5 R0 M128

Tool angle of inclination in the working plane coordinate system:
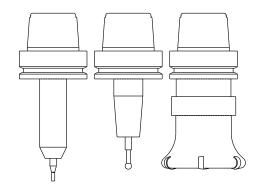
Example
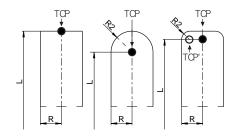
6	FUNCTION TCPM F TCP AXIS SPAT PATHCTRL AXIS
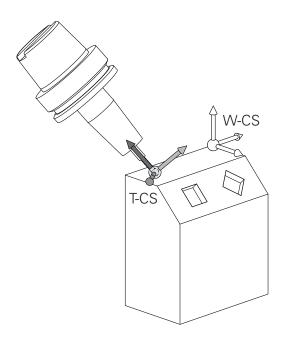
7	L A+0 B+45 C+0 R0 F2500

7	LN X+48 Y+102 Z-1.5 NX-0.04658107 NY0.00045007 NZ0.8848844 TX-0.08076201 TY-0.34090025 TZ0.93600126 R0 M128

7	LN X+48 Y+102 Z-1.5 NX-0.04658107 NY0.00045007 NZ0.8848844 R0 M128

> ⓘ With the shown positioning blocks with vectors, 3-D tool compensation is possible with compensation values **DL**, **DR** and **DR2** from the **TOOL CALL** block or from the **.tco** compensation table.
>
> The methods of function of the compensation values depend on the type of tool.
>
> The control detects the various tool types with the columns **L**, **R** and **R2** of the tool table:
>
> - $R2_{TAB} + DR2_{TAB} + DR2_{PROG} = 0$
>   → end mill
> - $R2_{TAB} + DR2_{TAB} + DR2_{PROG} = R_{TAB} + DR_{TAB} + DR_{PROG}$
>   → radius cutter or ball cutter
> - $0 < R2_{TAB} + DR2_{TAB} + DR2_{PROG} < R_{TAB} + DR_{TAB} + DR_{PROG}$
>   → toroid cutter or toroidal cutter
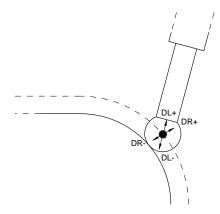
> ⓘ Without the **TCPM** function or miscellaneous function **M128**, orientation of the tool coordinate system and input coordinate system is identical.

3.8 Accessories: HEIDENHAIN 3-D touch probes and electronic handwheels

3-D touch probes

Applications for HEIDENHAIN 3-D touch probes:

- Automatically align workpieces
- Quickly and precisely set presets
- Measure the workpiece during program run
- Measure and inspect tools

> All of the cycle functions (touch probe cycles and fixed cycles) are described in the **Cycle Programming** User's Manual. If you need this User's Manual, please contact HEIDENHAIN.
> ID: 892905-xx

Touch trigger probes TS 260, TS 444, TS 460, TS 642 and TS 740

The TS 248 and TS 260 touch probes are particularly cost-effective and transmit the trigger signals via a cable.

The wireless TS 740 and TS 642 touch probes as well as the smaller TS 460 and TS 444 touch probes are suitable for use on machines with tool changers. All of the above touch probes feature infrared signal transmission. TS 460 also supports wireless transmission and offers optional collision protection. Thanks to an integrated air turbine generator, the TS 444 touch probe is battery-free.

HEIDENHAIN touch trigger probes feature either a wear-resistant optical switch or several high-precision pressure sensors (TS 740) that detect the deflection of the stylus. On deflection, a trigger signal is generated, which causes the control to store the current position of the touch probe as the actual value.

Tool touch probes TT 160 and TT 460

The TT 160 and TT 460 touch probes are designed for the efficient and precise measurement and inspection of tool dimensions.

The control offers cycles that enable you to determine the tool length and radius while the spindle is rotating or stationary. The tool touch probe features a particularly rugged design and a high degree of protection, which make it insensitive to coolants and swarf.

A wear-resistant optical switch generates the trigger signal. With the TT 160, signal transmission is by cable. The TT 460 supports infrared and radio transmission.

HR electronic handwheels

Electronic handwheels facilitate manual traversing of the axis slides. A wide range of traverses per handwheel revolution is available. The portable HR 510, HR 520 and HR 550FS handwheels are avaible from HEIDENHAIN in addition to the HR 130 and HR 150 panel-mounted handwheels.
Further information: "Traverse with electronic handwheels", Page 175

> ⚙ Several electronic handwheels can also be connected simultaneously and used alternatively on controls with the (**HSCI**: HEIDENHAIN Serial Controller Interface) serial interface for control components.
>
> Configuration is performed via the machine tool builder.

4

Tools

4.1 Tool data

Tool number, tool name

Each tool is identified by a number between 0 and 32767. If you are working with tool tables, you can also enter a tool name for each tool. Tool names can have up to 32 characters.

> ℹ️ **Permitted special characters**: # $ % & , - _ . 0 1 2 3 4 5 6 7 8 9 @ A B C D E F G H I J K L M N O P Q R S T U V W X Y Z
>
> The control automatically replaces lowercase letters with corresponding uppercase letters during saving.
>
> **Impermissible characters**: <blank space> ! " ' ( ) * + : ; < = > ? [/] ^ ` { | } ~

The tool number 0 is automatically defined as the zero tool with the length L=0 and the radius R=0. In tool tables, tool T0 should also be defined with L=0 and R=0.
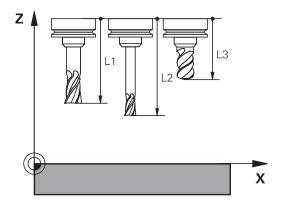
Tool length L

Always enter the tool length **L** as an absolute value based on the tool reference point.

> ℹ️ To determine the absolute tool length, the control requires various functions, such as the material removal simulation or **Dynamic Collision Monitoring (DCM)**.
>
> The absolute length of the touch probe always references the tool reference point. The machine tool builder usually defines the spindle nose as the tool reference point.

Measuring the tool length

Externally measure your tools with a tool presetter or directly in the machine, e.g. using a tool touch probe. If such measurements are not possible, you can determine the tool length.

You have the following options for determining the tool length:

- With a gauge block
- With a calibration pin (inspection tool)

> ℹ️ Before you determine tool length, you have to set the preset in the spindle axis.

Determining the tool length with a gauge block

> ℹ️ You can only set the preset with a gauge block if the tool reference point is at the spindle nose.
>
> Place the preset on the surface on which will then be touched-off with the tool. This surface might have to be created first.

Proceed as follows to set the datum with a gauge block:

- ▶ Place the gauge block on the machine table
- ▶ Position the spindle nose next to the gauge block
- ▶ Gradually move in **Z+** direction until you can just slide the gauge block under the spindle nose
- ▶ Set the datum in **Z**

To determine the tool length, proceed as follows:

- ▶ Insert the tool
- ▶ Touch off the surface
- > The control displays the absolute tool length as the actual position in the position display.

Determining the tool length with a calibration pin and a tool setter

Proceed as follows to set the preset with a calibration pin and a tool setter:
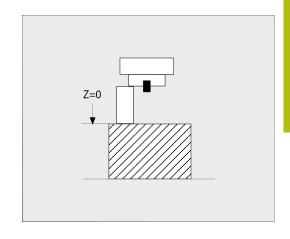
- ▶ Clamp the tool setter onto the machine table.
- ▶ Bring the flexible inner ring of the tool setter to the same height as the fixed outer ring.
- ▶ Set the gauge to 0
- ▶ Move the calibration pin onto the flexible inner ring.
- ▶ Set the datum in **Z**
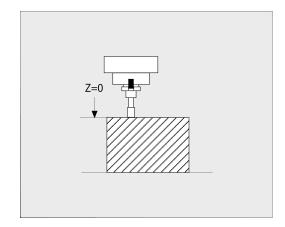
To determine the tool length, proceed as follows:

- ▶ Insert the tool
- ▶ Move the tool onto the flexible inner ring until the gauge displays 0.
- > The control displays the absolute tool length as the actual position in the position display.

Tool radius R

You can enter the tool radius R directly.

Tool table fundamentals

You can define and store up to 32 767 tools and their tool data in a tool table.

You must use tool tables if:

- you wish to use indexed tools such as stepped drills with more than one length compensation value
  Further information: "Indexed tool", Page 133
- If your machine has an automatic tool changer
- If you want to apply fine roughing with the fixed cycle 22
  Further information: Cycle Programming User's Manual
- If you want to work with fixed cycles 251 to 254
  Further information: Cycle Programming User's Manual

NOTICE

Caution: Data may be lost!

Deleting line 0 from the tool table will destroy the structure of the table. As a result, locked tools might no longer be recognized as locked and, consequently, the search for a replacement tool will not work, either. The problem cannot be solved by reinserting a line 0. The original tool table will be permanently damaged!

- ▶ Restore the tool table
  - Add a new line 0 to the defective tool table
  - Copy the defective tool table (e.g. toolcopy.t)
  - Delete the defective tool table (current tool.t)
  - Copy the copied tool table (toolcopy.t) as tool.t
  - Delete the copied tool table (toolcopy.t)
- ▶ Contact HEIDENHAIN Service (NC helpline)

All table names must start with a letter. Please keep this in mind when creating and managing additional tables.

You can select the table view with the **Screen Layout** key. You can choose between a list view and a form view.

Other settings, such as **SORT/ HIDE COLUMNS**, can be made after the file has been opened.

Indexed tool

Step drills, T-slot milling cutters, side milling cutters and, in general, all tools that require the input of multiple length and radius data cannot be fully defined in a single line of the tool table. Each line of the table permits the definition of one length and one radius.

In order to assign multiple compensation data to a tool (multiple tool table lines), add an indexed tool number (such as **T 5.1**) to an existing tool definition (**T 5**). Each additional line of the table thus comprises the original tool number, a period and an index (in ascending order from 1 to 9). The original tool table line contains the maximum tool length; the tool lengths in the subsequent table lines are given in descending order of their distance to the tool holder point.

Proceed as follows to create an indexed tool number (table line):

INSERT
LINE

▶ Open the tool table
▶ Press the **Insert Line** soft key
> The control opens the **Insert Line** pop-up window
▶ In the **Number of new lines =** input field, enter the number of lines to add
▶ Enter the original tool number into the **Tool number** input field
▶ Confirm with **OK**
> The control adds the additional lines to the tool table

ⓘ The **Dynamic Collision Monitoring (DCM)** function also uses the length and radius data for displaying the active tool and for collision monitoring. Incomplete or incorrect tool definitions may lead to premature or false collision warnings.

Quick search for the tool name:

If the **EDIT** soft key is set to **OFF**, you can search for a tool name. Proceed as follows:

▶ Enter the first few characters of the tool name, e.g. **MI**
> The control shows a dialog box with the entered text and jumps to the first match.
▶ Enter additional characters to narrow down the search result, e.g. **MILL**
▶ If the control cannot find any more matches for the entered search string, you can press the last entered character (e.g. **L**) to jump between matches, as with the arrow keys.

The quick search can also be used for tool selection in the **TOOL CALL** block.

Displaying only specific tool types (filter setting)

▶ Press the **TABLE FILTER** soft key

▶ Select the desired tool type by soft key

> The control displays only tools of the selected type.

▶ Cancel the filter: Press the **SHOW ALL** soft key

⚙   Refer to your machine manual!

        The machine tool builder adapts the features of the filter
        function to the requirements of your machine.

Soft key	Filter functions of the tool table
TABLE FILTER	Select the filter function
SHOW ALL	Cancel the filter settings and show all tools
DEFAULT FILTER	Use the default filter
DRILL	Show all drills in the tool table
CUTTER	Show all cutters in the tool table
THREADTOOL	Show all taps/thread cutters in the tool table
TCH. PROBE	Show all touch probes in the tool table

Hiding or sorting the tool table columns

You can adapt the layout of the tool table to your needs. Columns that are not to be displayed can be simply hidden:

▶ Press the **SORT/ HIDE COLUMNS** soft key

▶ Select the appropriate column name with the arrow key

▶ Press the **HIDE COLUMN** soft key to remove this column from the table view

You can also modify the sequence of columns in the table:

▶ You can also modify the sequence of columns in the table with the **Move before:** dialog. The entry highlighted in **Displayed columns:** is moved in front of this column

Use a connected mouse or the navigation keys to work in the form.

Proceed as follows:

▶ Press the navigation keys to go to the input fields

▶ Use the arrow keys to navigate within an input field

▶ Press the **GOTO** key to open drop-down menus

ⓘ The function **freeze number of columns** enables you to determine how many columns (0-3) the control will freeze to the left border of the screen. These columns will remain visible when you navigate to the right within the table.

Tool table for turning tools

When managing turning tools, other geometric descriptions than those for milling or drilling tools are required. To be able to execute tool radius compensation, for example, you have to define the tool radius. The control provides special tool management for turning tools to support this definition process.

Further information: "Tool data", Page 400

Tool table for grinding tools

When managing grinding tools, other geometric descriptions than those for milling or drilling tools are required. The control provides special form-based tool management for grinding and dressing tools.

Further information: "Tools for grinding (Option 156)", Page 413

Creating and activating an INCH-based tool table

ℹ️   If you choose to switch the units for your control to
      INCH, the tool table unit of measure will not be adjusted
      automatically.

      If you want to change the unit of measure in the table as
      well, you need to create a new tool table.

      This applies to all tool tables, including the **toolturn.trn**
      table for turning tools. You can repeat the steps below
      for other tool tables as well.

To create and activate an **INCH**-based tool table, proceed as
follows:

▢   ▶ Select **Positioning w/ Manual Data Input**
             operating mode
          ▶ Call the zero tool (T0)
          ▶ Restart the control
          ▶ Do **not** confirm **Power interrupted** with **CE**

◈   ▶ Select the **Programming** operating mode

PGM   ▶ Open the file manager
MGT
          ▶ Open the **TNC:\table** folder
          ▶ Rename the **tool.t** file, e.g. to **tool_mm.t**
          ▶ Create a new file and name it **tool.t**

INCH   ▶ Select the **INCH** unit of measure
          > The control opens the new, empty tool table.

APPEND   ▶ Append lines, e.g. 100 lines
N LINES
AT END   > The control adds the lines
          ▶ Position the cursor in column **L** of line **0**
          ▶ Enter **0**
          ▶ Position the cursor in column **R** of line **0**
          ▶ Enter **0**

ENT   ▶ Confirm your input

PGM   ▶ Open the file manager
MGT
          ▶ Open any NC program

🖐   ▶ Select **Manual operation** mode
          ▶ Confirm **Power interrupted** with **CE**

TOOL   ▶ Open the tool table
TABLE
          ▶ Review the tool table

ℹ️   The preset table is another table whose unit of measure
      is not adjusted automatically.
      Further information: "Creating and activating an INCH-
      based preset table", Page 195

Entering tool data into the table

Standard tool data

Abbr.	Inputs	Dialog
T	Number by which the tool is called in the NC program (e.g. 5, indexed: 5.2)	-
NAME	Name by which the tool is called in the NC program (max. 32 characters, all capitals, no spaces)	**Tool name?**
L	Tool length L	**Tool length?**
R	Tool radius R	**Tool radius?**
R2	Tool radius R2 for toroid cutters (only for 3-D radius compensation or graphical representation of a machining operation with Ball-nose cutter)	**Tool radius 2?**
DL	Delta value for tool length L	**Tool length oversize?**
DR	Delta value for tool radius R	**Tool radius oversize?**
DR2	Delta value for tool radius R2	**Tool radius oversize 2?**
TL	Set tool lock (**TL** for **T**ool**L**ocked)	**Tool locked? Yes=ENT/No=NOENT**
RT	Number of a replacement tool (**RT** for **R**eplacement**T**ool) An empty field or input **0** means there is no replacement tool	**Replacement tool?**
TIME1	Maximum tool life in minutes. This function can vary depending on the individual machine tool. Your machine manual provides more information	**Maximum tool age?**
TIME2	Maximum tool life in minutes during a tool call: If the current tool age reaches or exceeds this value, the control inserts the replacement tool during the next **TOOL CALL** (if the tool axis is specified)	**Max. tool age for TOOL CALL?**
CUR_TIME	Current age of the tool in minutes: The control automatically counts the current tool life (**CUR_TIME**: For **CUR**rent **TIME**). A starting value can be entered for used tools	**Current tool age?**
TYPE	Tool type: Press the **ENT** key to edit the field. The **GOTO** key opens a window in which you can select the tool type. In the tool management, use the **SELECT** soft key to open the pop-up window. You can assign tool types to specify the display filter settings such that only the selected type is visible in the table	**Tool type?**
DOC	Comment on tool (max. 32 characters)	**Tool description**
PLC	Information on this tool that is to be sent to the PLC	**PLC status?**
LCUTS	Tooth length of the tool for Cycles 22, 233, 256, 257	**Tooth length in the tool axis?**
ANGLE	Maximum plunge angle of the tool for reciprocating plunge-cut in Cycles 22 and 208	**Maximum plunge angle?**
TMAT	Cutting material of the tool for the cutting data calculator	**Tool material?**
CUTDATA	Cutting data table for the cutting data calculator	**Cutting data table?**

Abbr.	Inputs	Dialog
NMAX	Limit the spindle speed for this tool. The programmed value is monitored (error message) as well as an increase in the shaft speed via the potentiometer. Function inactive: Enter **-**. **Input range**: 0 to +999, if function not active: enter **-**	**Maximum speed [rpm]**
LIFTOFF	Definition of whether the control should retract the tool in the direction of the positive tool axis at an NC stop in order to avoid leaving dwell marks on the contour. If **Y** is defined, the control retracts the tool from the contour, provided **M148** has been activated. **Further information:** "Automatically retracting the tool from the contour at an NC stop: M148", Page 317	**Retract allowed? Yes=ENT/ No=NOENT**
TP_NO	Reference to the number of the touch probe in the touch-probe table	**Number of the touch probe**
T-ANGLE	Point angle of the tool. Is used by the Centering cycle (Cycle 240) in order to calculate the centering depth from the diameter entry	**Point angle**
PITCH	Thread pitch of the tool. Used by tapping cycles (Cycle 206, Cycle 207 and Cycle 209). A positive algebraic sign means a right-hand thread.	**Tool thread pitch?**
AFC	Control setting for the adaptive feed control from the **AFC.TAB**. In the tool table, use the **SELECT** soft key to open the selection list. In tool management, use the **SELECT** soft key and the **OK** soft key to confirm your selection. **Input range**: max. 10 characters	**Feedback-control strategy**
AFC-LOAD	Tool-dependent standard reference power for adaptive feed control AFC. The input in percent refers to the rated spindle power. The control immediately uses the value given for regulation, meaning a teach-in cut is dropped. The value should be previously determined with a teach-in cut. **Further information:** "Recording a teach-in cut", Page 336	**Reference power for AFC [%]**
AFC-OVLD1	Cut-related tool wear monitoring for the adaptive feed control. The input in percent refers to the standard reference power. The value 0 deactivates the monitoring function. An empty field has no effect. **Further information:** "Tool wear monitoring", Page 343	**AFC overload warning level [%]**
AFC-OVLD2	Cut-related tool load monitoring (tool breakage control) for the adaptive feed control. The input in percent refers to the standard reference power. The value 0 deactivates the monitoring function. An empty field has no effect. **Further information:** "Tool load monitoring", Page 343	**AFC ovrload switch-off level [%]**
LAST_USE	Date and time that the tool was last inserted via **TOOL CALL**	**Date/time of last tool call**

Abbr.	Inputs	Dialog
PTYP	Tool type for evaluation in the pocket table Refer to your machine manual! Function is defined by the machine tool builder!	**Tool type for pocket table?**
ACC	Activate or deactivate active chatter control for the respective tool (Page 344). **Input range**: N (inactive) and Y (active)	**ACC active? Yes=ENT/No=NOENT**
KINEMATIC	Use the **SELECT** soft key to show the tool carrier kinematics. In the tool management, use the **SELECT** soft key and the **OK** soft key to adopt the file name and the path **Further information:** "Allocating parameterized tool carriers", Page 166	**Tool-carrier kinematics**
DR2TABLE	Use the **SELECT** soft key to show the list of error compensation tables and select the error compensation table (without extension and path). The error compensation tables are saved under TNC:\system\3D-ToolComp.	**Compensation val. table for DR2**
OVRTIME	Time for exceeding the tool life in minutes **Further information:** "Overtime for tool life", Page 149 Refer to your machine manual! Function is defined by the machine tool builder!	**Tool life expired**

Tool data for automatic tool measurement

> ⚙ Refer to your machine manual!
>
> The machine tool builder defines whether the **R-OFFS** offset will be taken into account for a tool with **CUT** 0.
>
> Your machine tool builder defines the default value for the **R-OFFS** and **L-OFFS** columns.

Abbr.	Inputs	Dialog
CUT	Number of teeth (99 teeth maximum)	**Number of teeth?**
LTOL	Permissible deviation from tool length L for wear detection. If the entered value is exceeded, the control locks the tool (status **L**). Input range: 0 to 0.9999 mm	**Wear tolerance: length?**
RTOL	Permissible deviation from tool radius R for wear detection. If the entered value is exceeded, the control locks the tool (status **L**). Input range: 0 to 0.9999 mm	**Wear tolerance: radius?**
R2TOL	Permissible deviation from tool radius R2 for wear detection. If the entered value is exceeded, the control locks the tool (status **L**). Input range: 0 to 0.9999 mm	**Wear tolerance: Radius 2?**
DIRECT	Cutting direction of the tool for measuring the tool during rotation	**Cutting direction? M4=ENT/ M3=NOENT**
R-OFFS	Tool length measurement: Tool offset between stylus center and tool center.	**Tool offset: radius?**
L-OFFS	Radius measurement: Tool offset between upper surface of stylus and lower surface of tool in addition to **offset-ToolAxis**.	**Tool offset: length?**
LBREAK	Permissible deviation from tool length **L** for breakage detection. If the entered value is exceeded, the control locks the tool (status **L**). Input range: 0 to 3.2767 mm	**Breakage tolerance: length?**
RBREAK	Permissible deviation from tool radius R for breakage detection. If the entered value is exceeded, the control locks the tool (status **L**). Input range: 0 to 0.9999 mm	**Breakage tolerance: radius?**

> 📖 For a description of the cycles governing automatic tool measurement,
>
> **Further information:** Cycle Programming User's Manual

Editing the tool tables

The tool table that is active during execution of the part program is designated TOOL.T and must be saved in the **TNC:\table** directory.

Other tool tables that are to be archived or used for test runs are given different file names with the extension .T. By default, for the **Test Run** and **Programming** modes the control also uses the TOOL.T tool table. In the **Test Run** mode, press the **TOOL TABLE** soft key to edit it.

To open the tool table TOOL.T:

▶ Select any machine operating mode

| TOOL TABLE | ▶ Select the tool table: Press the **TOOL TABLE** soft key |

| EDIT OFF ON | ▶ Set the **EDIT** soft key to **ON** |
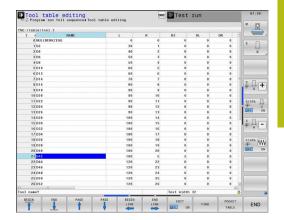
> 🛈 If you edit the tool table, the selected tool is locked. If this tool is required in the NC program being used, the control shows the message: **Tool table locked**.
>
> If a new tool is created, the length and radius columns remain empty until entered manually. An attempt to insert such a newly created tool will be aborted by the control and an error message will appear. This means you cannot insert a tool for which no geometry data are available yet.

Proceed as follows to use the alphabetic keyboard or a connected mouse for navigation and editing:

- Arrow keys: move from one cell to the next
- ENT key: jump to the next cell; with selection fields: open the selection dialog
- Mouse click on a cell: move to the cell
- Double click on a cell: place the cursor in the cell; with selection fields: open the selection dialog

Soft key	Editing functions of the tool table
BEGIN	Select the table start
END	Select the table end
PAGE	Select the previous page in the table
PAGE	Select the next page in the table
FIND	Find the text or number
BEGIN LINE	Go to beginning of line

Soft key	Editing functions of the tool table
END LINE	Go to end of line
COPY FIELD	Copy active field
PASTE FIELD	Insert copied field
APPEND N LINES AT END	Add the entered number of lines (tools) at the end of the table
INSERT LINE	Insert a line with definable tool number
DELETE LINE	Delete the current line (tool)
SORT	Sort the tools according to the content of a column
SELECT	Select possible entries from a pop-up window
RESET COLUMN	Reset the value
EDIT CURRENT FIELD	Place the cursor in the current cell

Importing tool tables

> Refer to your machine manual!
>
> The machine tool builder can adapt the **ADAPT NC PGM / TABLE** function.
>
> The machine tool builder can define update rules that make it possible, for example, to automatically remove umlauts from tables and NC programs.

If you export a tool table from an iTNC 530 and import it into a TNC 640, you have to adapt its format and content before you can use the tool table. On the TNC 640, you can adapt the tool table conveniently with the **ADAPT NC PGM / TABLE** function. The control converts the contents of the imported tool table to a format valid for the TNC 640 and saves the changes to the selected file.

Proceed as follows:

▶ Save the tool table of the iTNC 530 to the **TNC:\table** directory

⟳   ▶ Select the **Programming** operating mode

PGM
MGT   ▶ Press the **PGM MGT** key

↑   ▶ Move the cursor to the tool table you want to import

MORE
FUNCTIONS   ▶ Press the **MORE FUNCTIONS** soft key

ADAPT
NC PGM /
TABLE   ▶ Press the **ADAPT NC PGM / TABLE** soft key

    > The control asks you whether you want to overwrite the selected tool table.

    ▶ Press the **CANCEL** soft key

    ▶ Alternative: Press the **OK** soft key to overwrite

    ▶ Open the converted table and check its contents

    > New columns in the tool table are highlighted green.

    ▶ Press the **REMOVE UPDATE INFO** soft key

    > The green columns are displayed in white again.

ⓘ   The following characters are permitted in the **Name** column of the tool table: # $ % & , - . 0 1 2 3 4 5 6 7 8 9 @ A B C D E F G H I J K L M N O P Q R S T U V W X Y Z _

During the import, a comma is converted to a period.

The control overwrites the active tool table when importing an external table with the same name. To prevent data loss, back up the original tool table before you start the import!

The procedure for copying tool tables using the file manager is described in the section on file management.

Further information: User's Manual for Conversational Programming or ISO Programming

When iTNC 530 tool tables are imported, all defined tool types are transferred as well. Tool types not present are imported as type **Undefined**. Check the tool table after the import.

Overwriting tool data from an external PC

Application

The **TNCremo** software provides an especially convenient way to use an external PC to overwrite tool data.

Further information: "Software for data transfer", Page 483

This application case occurs if you wish to determine tool data on an external tool presetter and then transfer this to the control.

Requirements

In addition to Option 18, HEIDENHAIN DNC, **TNCremo** (version 3.1 or higher) is required. During the installation, the **TNCremoPlus** function must be selected.

Procedure
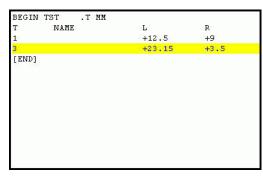
▶ Copy the TOOL.T tool table to the control, for example to TST.T

▶ Start the **TNCremo** data transfer software on the PC

▶ Connect to the control

▶ Transfer the copied tool table TST.T to the PC

▶ Use any text editor to reduce TST.T to the lines and columns to be changed (see figure). Make sure that the header is not changed and the data is always flush in the column. The tool number (column T) need not be consecutive

▶ In **TNCremo**, select the menu item <Extras> and <TNCcmd>: This starts TNCcmd

▶ To transfer the TST.T file to the control, enter the following command and confirm with the return key (see figure): put tst.t tool.t /m
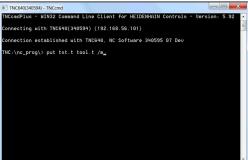
ℹ   During transfer, only the tool data defined in the subfile (e.g. TST.T) is overwritten. All other tool data of the table TOOL.T remains unchanged.

The procedure for copying tool tables using the file manager is described in the file management.

Further information: User's Manual for Conversational Programming or ISO Programming
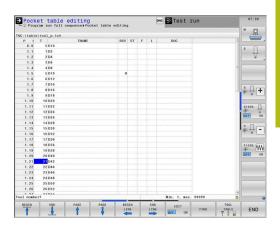
Pocket table for tool changer

⚙️ Refer to your machine manual!

The machine tool builder adapts the features of the pocket table to the requirements of your machine.

You need a pocket table for automatic tool changing. You manage the assignment of your tool changer in the pocket table. The pocket table is in the **TNC:\table** directory. The machine manufacturer can amend the name, path and content of the pocket table. If you wish, you can select different views using the soft keys in the **TABLE FILTER** menu.

Editing a pocket table in a Program Run operating mode

| TOOL TABLE | ▶ Select the tool table: Press the **TOOL TABLE** soft key |

| POCKET TABLE | ▶ Press the **POCKET TABLE** soft key |

| EDIT OFF ON | ▶ If required, set the **EDIT** soft key to **ON** |

Selecting a pocket table in Programming mode

Proceed as follows to select the pocket table in the Programming operating mode:

▶ To call the file manager, press the **PGM MGT** key.

▶ Press the **SHOW ALL** soft key

▶ Select a file or enter a new file name

▶ Confirm your entry with the **ENT** key or the **SELECT** soft key

Abbr.	Inputs	Dialog
P	Pocket number of the tool in the tool magazine	-
T	Tool number	**Tool number?**
RSV	Pocket reservation for box magazines	**Pocket reserv.: Yes = ENT / No = NOENT**
ST	Special tool (**ST**); If your special tool blocks pockets in front of and behind its actual pocket, these additional pockets need to be locked in column L (status L).	**Special tool?**
F	The tool is always returned to the same pocket in the tool magazine	**Fixed pocket? Yes = ENT / No = NO ENT**
L	Locked pocket (**L**: for **L**ocked)	**Pocket locked Yes = ENT / No = NO ENT**
DOC	Display of the comment to the tool from TOOL.T	-
PLC	Information on this tool pocket that is to be sent to the PLC	**PLC status?**
P1 ... P5	Function is defined by the machine tool builder. The machine tool documentation provides further information	**Value?**
PTYP	Tool type. Function is defined by the machine tool builder. The machine tool documentation provides further information	**Tool type for pocket table?**
LOCKED_ABOVE	Box magazine: Lock the pocket above	**Lock the pocket above?**
LOCKED_BELOW	Box magazine: Lock the pocket below	**Lock the pocket below?**
LOCKED_LEFT	Box magazine: Lock the pocket at left	**Lock the pocket at left?**
LOCKED_RIGHT	Box magazine: Lock the pocket at right	**Lock the pocket at right?**

Soft key	Editing functions for pocket tables
BEGIN ↑	Select the table start
END ↓	Select the table end
PAGE ↑	Select the previous page in the table
PAGE ↓	Select the next page in the table
RESET POCKET TABLE	Reset pocket table Depends on optional machine parameter **enableReset** (no.106102)
RESET COLUMN T	Reset tool number T column Depends on optional machine parameter **showResetColumnT** (no. 125303)
BEGIN LINE ←	Go to beginning of line
END LINE →	Go to end of line
SIMULATED TOOL CHANGE	Simulate a tool change
SELECT	Select a tool from the tool table: The control shows the contents of the tool table. Use the arrow keys to select a tool, press **OK** to transfer it to the pocket table
RESET COLUMN	Reset the value
EDIT CURRENT FIELD	Place the cursor in the current cell
SORT	Sort the view

⚙ Refer to your machine manual!

The machine manufacturer defines the features, properties and designations of the various display filters.

Tool change

Automatic tool change

⚙ Refer to your machine manual!
The tool change function can vary depending on the individual machine tool.

If your machine tool has automatic tool changing capability, the program run is not interrupted. When the control reaches a tool call with **TOOL CALL**, it replaces the inserted tool by another from the tool magazine.

Automatic tool change if the tool life expires: M101

⚙ Refer to your machine manual!
The function of **M101** can vary depending on the individual machine tool.

When the specified tool life has expired, the control can automatically insert a replacement tool and continue machining with it. Activate the miscellaneous function **M101** for this. **M101** is reset with **M102.**

Enter the respective tool life after which machining is to be continued with a replacement tool in the **TIME2** column of the tool table. In the **CUR_TIME** column the control enters the current tool life.

If the current tool life is higher than the value entered in the **TIME2** column, a replacement tool will be inserted at the next possible point in the program no later than one minute after expiration of the tool life. The change is made only after the NC block has been completed.

NOTICE

Danger of collision!

During an automatic tool change with **M101**, the control always retracts the tool in the tool axis first. There is danger of collision when retracting tools for machining undercuts, such as side milling cutters or T-slot milling cutters!

▶ Deactivate the tool change with **M102**

After the tool change the control positions the tool according to the following logic, unless otherwise specified by the machine tool builder:

- If the target position in the tool axis is below the current position, the tool axis is positioned last
- If the target position in the tool axis is above the current position, the tool axis is positioned first

Prerequisites for a tool change with M101

> ℹ️ As replacement tools, use only tools with the same radius. The control does not automatically check the radius of the tool.
>
> If you want the control to check the radius of the replacement tool, enter **M108** in the NC program.

The control performs the automatic tool change at a suitable point in the program. The automatic tool change is not performed:

- During execution of fixed cycles
- While radius compensation (**RR/RL**) is active
- Directly after an approach function **APPR**
- Directly before a departure function **DEP**
- Directly before and after **CHF** and **RND**
- During execution of macros
- During execution of a tool change
- Directly after a **TOOL CALL** or **TOOL DEF**
- During execution of SL cycles

Overtime for tool life

> ⚙️ This function must be enabled and adapted by the machine tool builder.

The tool condition at the end of planned tool life depends on e.g. the tool type, machining method and workpiece material. In the **OVRTIME** column of the tool table, enter the time in minutes for which the tool is permitted to be used beyond the tool life.

The machine manufacturer specifies whether this column is enabled and how it is used during tool search.

Tool usage test

Requirements

> ⚙️ Refer to your machine manual!
> The tool usage test function must be enabled by your machine tool builder.

> ℹ️ The tool usage test function is not available for turning tools.

To conduct a tool usage test, you must activate **Create tool usage files** in the MOD menu.

Further information: "Generating a tool usage file", Page 435

Generating a tool usage file

Depending on the setting in the MOD menu, you have the following options for generating the tool usage file:

- Completely simulate the NC program in the **Test Run** operating mode
- Completely run the NC program in the **Program Run, Full Sequence/Single Block** operating modes
- In the **Test Run** operating mode, press the **GENERATE TOOL USAGE FILE** soft key (also possible without simulation)

The tool usage file generated is in the same directory as the NC program. It contains the following information:

Column	Meaning
TOKEN	- **TOOL**: Tool usage time per tool call. The entries are listed in chronological order. - **TTOTAL**: Total usage time of a tool - **STOTAL**: Call of a subprogram. The entries are listed in chronological order. - **TIMETOTAL**: The total machining time of the NC program is entered in the **WTIME** column. In the **PATH** column the control saves the path name of the corresponding NC program. The **TIME** column shows the sum of all **TIME** entries (feed time without rapid traverse movements). The control sets all other columns to 0 - **TOOLFILE**: In the **PATH** column, the control saves the path name of the tool table with which you conducted the test run. This enables the control during the actual tool usage test to detect whether you performed the test run with TOOL.T
TNR	Tool number (**-1**: Tool not inserted yet)
IDX	Tool index
NAME	Tool name from the tool table
TIME	Tool usage time in seconds (feed time without rapid traverse movements)
WTIME	Tool-usage time in seconds (total usage time between tool changes)
RAD	**Tool radius R + Oversize of tool radius DR** from the tool table. (in mm)
BLOCK	Block number in which the **TOOL CALL** block was programmed
PATH	- **TOKEN** = **TOOL**: Path name of the active main program or subprogram - **TOKEN** = **STOTAL**: Path name of the subprogram
T	Tool number with tool index

Column	Meaning
OVRMAX	Maximum feed rate override that occurred during machining. The control enters the value 100 (%) during the test run
OVRMIN	Minimum feed rate override that occurred during machining. The control enters the value -1 during the test run
NAMEPROG	■ 0: The tool number is programmed ■ 1: The tool name is programmed

The control saves the tool usage times in a separate file with the extension **pgmname.H.T.DEP**. This file is not visible unless the machine parameter **dependentFiles** (no. 122101) is set to **MANUAL**

There are two ways to run a tool usage test for a pallet file:

■ If the cursor in the pallet file is on a pallet entry, the control runs the tool usage test for the entire pallet.

■ If the cursor in the pallet file is on a program entry, the control runs the tool usage test only for the selected NC program.

Using a tool usage test

Before starting a program in the **Program Run, Full Sequence/ Single Block** operating modes, you can check whether the tools being used in the selected NC program are available and have sufficient remaining tool life. The control then compares the actual tool-life values in the tool table with the nominal values from the tool usage file.

TOOL USAGE	▶ Press the **TOOL USAGE** soft key
TOOL USAGE TEST	▶ Press the **TOOL USAGE TEST** soft key > The control opens the **Tool usage test** pop-up window indicating the result of the usage test.
OK	▶ Press the **OK** soft key > The control closes the pop-up window.
ENT	▶ Alternative: Press the **ENT** key
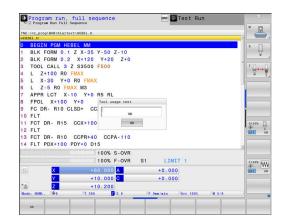
You can query the tool usage test with the **FN 18 ID975 NR1** function.

4.2 Tool management

Basics

⚙ Refer to your machine manual!

Tool management is a machine-dependent function, which may be partly or completely deactivated.
The machine tool builder defines the exact range of functions.

In tool management, your machine manufacturer can provide a wide range of functions for tool handling. Examples:
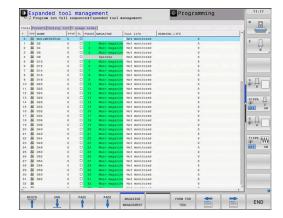
- Display and editing of all tool data from the tool table, the turning tool table and the touch probe table
- Easily readable and adaptable representation of the tool data in fillable forms
- Any description of the individual tool data in the new table view
- Mixed representation of data from the tool table and the pocket table
- Fast sorting of all tool data by mouse
- Use of graphic aids, e.g. color coding of tool or magazine status
- Copying and pasting of all tool data pertaining to a tool
- Graphic depiction of tool type in the table view and in the detail view for a better overview of the available tool types

The Expanded Tool Management (option 93) additionally provides:

- Provide a program-specific or pallet-specific usage order of all tools
- Provide a program-specific or pallet-specific tooling list of all tools

ℹ If you edit a tool in tool management, the selected tool is locked. If this tool is required in the NC program being used, the control shows the message: **Tool table locked**.

Calling tool management

⚙ Refer to your machine manual!
   The procedure for calling the tool management may
   differ from that described below.
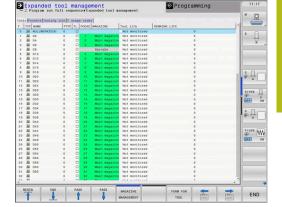
[TOOL TABLE] ▶ Select the tool table: Press the **TOOL TABLE** soft
                key

[▷]   ▶ Scroll through the soft-key row

[TOOL MANAGEMENT] ▶ Press the **TOOL MANAGEMENT** soft key
                   ⊳ The control switches to the new table view.

Tool management view

In the new view, the control presents all tool information in the
following four tabs:

■ **Tools**: Tool-specific information
■ **pockets**: Pocket-specific information

The Expanded Tool Management (option 93) additionally provides:

■ **Tooling list**: List of all tools in the NC program that is selected in
  the Program Run mode (only if you have already created a tool
  usage file)
  Further information: "Tool usage test", Page 149
■ **T usage order**: List of the sequence of all tools that are inserted
  in the NC program selected in the Program Run mode (only if
  you have already created a tool usage file)
  Further information: "Tool usage test", Page 149
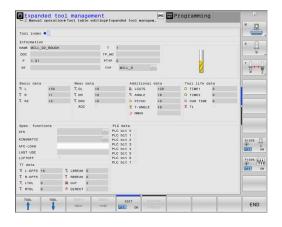
ℹ If a pallet table is selected in the Program Run operating
   mode, the **Tooling list** and **T usage order** are calculated
   for the entire pallet table.

Editing tool management

The tool management can be operated by mouse or with the keys and soft keys:

Soft key	Editing functions for tool management
BEGIN	Select the table start
END	Select the table end
PAGE	Select the previous page in the table
PAGE	Select the next page in the table
FORM FOR TOOL	Call the form view of the marked tool. Alternative function: Press the **ENT** key
	Changing tab: **Tools** and **pockets** Option 93 additionally provides: **Tooling list** and **T usage order**
FIND	Search function: Here you can select the column to be searched and then the search term either from a list or by entering it
TOOL IMPORT	Import tools
EXPORT TOOL	Export tools
DELETE MARKED TOOLS	Delete marked tools
APPEND N LINES AT END	Add several lines at end of table
UPDATE THE VIEW	Update table view
PROG. TOOL DISPLAY HIDE	Show the programmed tools column (if the **Pockets** tab is active)
COLUMN SORT MOVE	Define the settings: ■ **SORT COLUMN** active: Click the column header to sort the content of the column ■ **SHIFT COLUMN** active: The column can be moved by drag and drop
RESET SETTINGS	Reset the manually changed settings (move columns) to the original condition

i You can edit the tool data only in the form view. To activate the form view, press the **FORM FOR TOOL** soft key or the **ENT** key for the currently highlighted tool.

If you use the tool management without a mouse, then you can activate and deactivate functions with the **-/+** key.

In the tool management, use the **GOTO** key to search for the tool number or pocket number.

In addition, you can perform the following functions by mouse:

- Sorting function: You can sort the data in ascending or descending order (depending on the active setting of the soft key) by clicking a column of the table head.
- Arrange columns. You can arrange the columns in any sequence you want by clicking a column of the table head and then moving it with the mouse key pressed down. The control does not save the current column sequence when you exit the tool management (depending on the active setting of the soft key).
- Show miscellaneous information in the form view: The control displays tool tips when you leave the mouse pointer on an active entry field for more than a second and when you have set the **EDIT ON/OFF** soft key to **ON**

Editing with active form view

If the form view is active, the following functions are available to you:

Soft key	Editing functions, form view
TOOL ↑	Select the tool data of the previous tool
TOOL ↓	Select the tool data of the next tool
INDEX ←	Select previous tool index (only active if indexing is enabled)
INDEX →	Select the next tool index (only active if indexing is enabled)
SELECT	Open a pop-up window for selection (only available for selection fields)
DISCARD CHANGES	Discard all changes made since the form was called
CALCULATE TOOL COMPENSTN.	Calculate the measured values of tool compensation (only active for turning tools)
INSERT INDEX	Add tool index
DELETE INDEX	Delete tool index
COPY DATA RECORD	Copy the tool data of the selected tool
INSERT DATA REC.	Insert the copied tool data in the selected tool

Deleting marked tool data

Using this function you can simply delete tool data that you no longer need.

Follow the steps outlined below for deleting:

▶ In the tool management you use the arrow keys or mouse to mark the tool data that you wish to delete

▶ Press the **DELETE MARKED TOOLS** soft key

> The control shows a pop-up window listing the tool data to be deleted.

▶ Press the **START** soft key to start the deletion procedure

> The control shows a pop-up window with the deletion status.

▶ Terminate the delete process by pressing the **END** key or soft key

---

NOTICE

Caution: Data may be lost!

The **DELETE MARKED TOOLS** function permanently deletes the tool data. The data is not automatically backed up by the control, e.g. to a recycle bin, before being deleted. The data is irreversibly deleted by this function.

▶ Regularly back up important data to external drives

ℹ The tool data of tools still stored in the pocket table cannot be deleted. The tools must be removed from the magazine first.

Available tool types

ℹ In tool management, the control only displays the input fields related to the selected tool type.

The tool management displays the various tool types with an icon. The following tool types are available:

Icon	Tool type	Tool type number
	Undefined,****	99
	Milling cutter,MILL	0
	Roughing cutter,MILL_R	9
	Finishing cutter,MILL_F	10
	Ball-nose cutter, BALL	22
	Toroid cutter, TORUS	23
	Drill,DRILL	1

Icon	Tool type	Tool type number
	Tap, TAP	2
	Center drill, CENT	4
	Turning Tool, TURN	29
	Touch probe, TCHP	21
	Ream, REAM	3
	Countersink, CSINK	5
	Piloted counterbore, TSINK	6
	Boring tool, BOR	7
	Back boring tool, BCKBOR	8
	Thread mill, GF	15
	Thread mill w/ countersink, GSF	16
	Thread mill w/ single thread, EP	17
	Thread mill w/ indxbl insert, WSP	18
	Thread milling drill, BGF	19
	Circular thread mill, ZBGF	20
	Grinding tool	30
	Dressing tool	31

Importing and exporting tool data

Importing tool data

> ⚙ Refer to your machine manual!
>
> The machine tool builder can define update rules that make it possible, for example, to automatically remove umlauts from tables and NC programs.

Using this function you can simply import tool data that you have measured externally on a presetting device, for example. The file to be imported must have the CSV format (**c**omma **s**eparated **v**alue). **CSV** files are text files designed for the exchange of data with a simple structure. Accordingly, the import file must have the following structure:

- **Line 1**: In the first line you define the column names in which the data defined in the subsequent lines is to be placed. The column names are separated with a comma.
- **Other lines**: All the other lines contain the data that you wish to import into the tool table. The order of the data must match the order of the column names in Line 1. The data is separated by commas, decimal numbers are to be defined with a decimal point.

Follow the steps outlined below for importing:

- ▶ Copy the tool table to be imported to the **TNC:\system\tooltab** directory on the hard disk of the control
- ▶ Start expanded tool management
- ▶ Press the **TOOL IMPORT** soft key in the tool management
- > The control shows a pop-up window with the CSV files that are saved in the **TNC:\system\tooltab** directory
- ▶ Use the arrow keys or mouse to select the file to be imported and confirm with the **ENT** key
- > The control shows a pop-up window with the content of the CSV file
- ▶ Start the import procedure with the **EXECUTE** soft key.

i ■ The CSV file to be imported must be stored in the **TNC:\system\tooltab** directory.
   ■ If you import the tool data of existing tools (whose numbers are in the pocket table) the control issues an error message. You can then decide whether to skip this data record or insert a new tool. The control inserts a new tool into the first empty line of the tool table.
   ■ If the imported CSV file contains unknown table columns, the control displays a message during import. An additional note informs you that the data will not be transferred.
   ■ Make sure that the column designations have been specified correctly.
   Further information: "Entering tool data into the table", Page 137
   ■ You can import any tool data, the associated data record does not have to contain all the columns (or data) of the tool table.
   ■ The column names can be in any order, the data must be defined in the corresponding order.

Example

T,L,R,DL,DR	Line 1 with column names
4,125.995,7.995,0,0	Line 2 with tool data
9,25.06,12.01,0,0	Line 3 with tool data
28,196.981,35,0,0	Line 4 with tool data

Exporting tool data

Using this function you can simply export tool data to read it into the tool database of your CAM system, for example. The control stores the exported file in the CSV format (**c**omma **s**eparated **v**alue). **CSV** files are text files designed for the exchange of data with a simple structure. The export file has the following structure:

- **Line 1**: In the first line the control stores the column names of all the relevant tool data to be defined. The column names are separated from each other by commas.
- **Further lines**: All the other lines contain the data of the tools that you have exported. The order of the data matches the order of the column names in Line 1. The data is separated by commas, the control outputs decimal numbers with a decimal point.

Follow the steps outlined below for exporting:

- ► In the tool management you use the arrow keys or mouse to mark the tool data that you wish to export
- ► Press the **EXPORT TOOL** soft key
- > The control shows a pop-up window
- ► Enter a name for the CSV file and confirm it with the **ENT** key
- ► Start the export procedure with the **EXECUTE** soft key
- > The control shows a pop-up window with the status of the export process
- ► Terminate the export process by pressing the **END** key or soft key

---

❶ By default the control stores the exported CSV file in the **TNC:\system\tooltab** directory.

4.3   Tool carrier management

Fundamentals

You can create and manage tool carriers using the tool carrier management. The control factors the tool carriers into the calculations.

On machines with 3 axes, tool carriers for right-angled angle heads help processing on tool axes **X** and **Y**, as the control takes the dimensions of the angle heads into consideration.

Along with software option number 8, **Advanced Function Set** 1, you can tilt the working plane to the angle of the removable angled heads and thus keep working with the **Z** tool axis.

Along with software option number 40, **Dynamic Collision Monitoring**, you can monitor every tool carrier and thus prevent collisions.

You must carry out the following steps so that the control can factors the tool carriers into the calculations:

- Save tool carrier templates
- Assign input parameters to tool carriers
- Allocate parameterized tool carriers

Saving tool carrier templates

Many tool carriers only differ from others in terms of their dimensions, but their geometric shape is identical. So that you don't have to design all your tool carriers yourself, HEIDENHAIN supplies a range of ready-made tool carrier templates. Tool carrier templates are 3-D models with fixed geometries but changeable dimensions.

The tool carrier templates must be saved in **TNC:\system \Toolkinematics** and have the extension **.cft**.

🛈	If the tool carrier templates are not available in your control, please download the data you require from: **http://www.klartext-portal.com/nc-solutions/en**

🛈	If you need further tool carrier templates, please contact your machine manufacturer or third-party vendor.

🛈	The tool carrier templates may consist of several sub-files. If the sub-files are incomplete, the control will display an error message. **Do not use incomplete tool carrier templates!**

Assigning input parameters to tool carriers

Before the control can factor the tool carrier into the calculations, you must give the tool carrier template the actual dimensions. These parameters are entered in the additional **ToolHolderWizard** tool.

Save the parameterized tool carriers with the extension **.cfx** under **TNC:\system\Toolkinematics**.
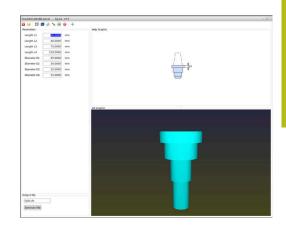
The additional **ToolHolderWizard** tool is mainly operated with a mouse. Using the mouse, you can also set the desired screen layout by drawing a line between the areas **Parameter**, **Help graphics** and **3-D graphics** by holding down the left mouse button.

The following icons are available in the additional **ToolHolderWizard** tool:

Icon	Function
✖	Close tool
🗁	Open file
⬚	Switch between wire frame model and solid object view
⬚	Switch between shaded and transparent view
⌞⌟	Display or hide transformation vectors
ᴬBᴄC	Show or hide names of collision objects
⊞	Display or hide test points
⊕	Show or hide measurement points
✛	Return to starting view of the 3-D model

> 🛈 If the tool carrier template does not contain any transformation vectors, names, test points and measurement points, the additional **ToolHolderWizard** tool does not execute any function when the corresponding icons are activated.

Parameterize the tool carrier template in the Manual operation mode

Proceed as follows to parameterize tool carrier templates and save these parameters:

▶ Press the **Manual operation** key

▶ Press the **TOOL TABLE** soft key

▶ Press the **EDIT** soft key

▶ Move the cursor to the **KINEMATIC** column

▶ Press the **SELECT** soft key

▶ Press the **TOOL HOLDER WIZARD** soft key
> The control opens the additional **ToolHolderWizard** tool in a pop-up window.

▶ Press the **OPEN FILE** icon
> The control opens a pop-up window.
▶ Select the desired tool carrier template using the preview screen
▶ Press the **OK** button
> The control opens the selected tool carrier template.
> The cursor goes to the first parameterizable value.
▶ Adjust values
▶ Enter the name for the parameterized tool holder in the **Output file** area
▶ Press the **GENERATE FILE** button
▶ If required, reply to the message on the control
▶ Press the **CLOSE** icon
> The control closes the additional tool

Parameterize the tool carrier template in the Programming operating mode

Proceed as follows to parameterize tool carrier templates and save these parameters:

▶ Press the **Programming** key

▶ Press the **PGM MGT** key
▶ Select the path **TNC:\system\Toolkinematics**
▶ Select the tool carrier template
> The control opens the additional **ToolHolderWizard** tool with the selected tool carrier template.
> The cursor goes to the first parameterizable value.
▶ Adjust values
▶ Enter the name for the parameterized tool holder in the **Output file** area
▶ Press the **GENERATE FILE** button
▶ If required, reply to the message on the control
▶ Press the **CLOSE** icon
> The control closes the additional tool

Allocating parameterized tool carriers

To allow the control to factor a parameterized tool carrier into calculations, you must allocate the tool carrier to a tool and **call the tool again**.

> ℹ️ Parameterized tool carriers can consist of several sub-files. If the sub-files are incomplete, the control will display an error message.
>
> **Only use fully parameterized tool carriers!**

Proceed as follows to allocate a parameterized tool carrier to a tool:

🖐	▶ Operating mode: Press the **Manual operation** key
TOOL TABLE	▶ Press the **TOOL TABLE** soft key
EDIT OFF ON	▶ Press the **EDIT** soft key
→	▶ Move the cursor to the **KINEMATIC** column of the required tool
SELECT	▶ Press the **SELECT** soft key
	> The control opens a pop-up window with parameterized tool carriers
	▶ Select the desired tool carrier using the preview screen
	▶ Press the **OK** soft key
	> The control copies the name of the selected tool carrier to the **KINEMATIC** column
END	▶ Exit the tool table
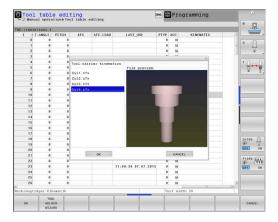
5

Setup

5.1 Switch-on, switch-off

Switch-on

⚠ **DANGER**

Caution: Danger for the operator!

Machines and machine components always present mechanical hazards. Electric, magnetic or electromagnetic fields are particularly hazardous for persons with cardiac pacemakers or implants. The hazard starts when the machine is powered up!

▶ Read and follow the machine manual
▶ Read and follow the safety precautions and safety symbols
▶ Use the safety devices

⚙ Refer to your machine manual!
  Switching on the machine and traversing the reference points can vary depending on the machine tool.

Switch the machine and the control on as follows:
▶ Switch on the power supply for the control and the machine
> The control displays the switch-on status in the subsequent dialogs.
> If booting was successful, the control displays the **Power interrupted** dialog

[CE]   ▶ Press the **CE** key to clear the message
       > The control displays the **Compiling PLC program** dialog; the PLC program is compiled automatically
       > The control displays the **Switch on external dc voltage** dialog

(I)   ▶ Switch on the machine control voltage
       > The control carries out a self-test.

If the control does not register an error, it displays the **Traverse reference points** dialog.

If the control registers an error, it issues an error message.

NOTICE

Danger of collision!

When the machine is switched on, the control tries to restore the switch-off status of the tilted plane. This is prevented under certain conditions. For example, this applies if axis angles are used for tilting while the machine is configured with spatial angles, or if you have changed the kinematics.

▶ If possible, reset the tilted condition before switching the machine off
▶ Check the tilted condition when switching the machine back on

---

Checking the axis positions

⚙ This section applies only to machine axes with EnDat encoders.

If the actual axis position after the machine is switched on does not match the position at switch-off, the control displays a pop-up window.

► Check the axis position of the affected axis
► If the current axis position matches that proposed in the display, confirm with **YES**

NOTICE

Danger of collision!

If they are not paid attention to, deviations between the actual axis positions and those expected by the control (saved at the time of switch-off) can lead to undesirable and unforeseeable movements of the axes. There is risk of collision during referencing and all subsequent movements.

► Check the axis positions
► Only confirm the pop-up window with **YES** if the axis positions match
► Despite confirmation, at first only move the axis carefully
► If there are discrepancies or you have any doubts, contact your machine tool builder

---

Traversing reference points

If the control performs the self-test successfully, it then displays the **Traverse reference points** dialog.

⚙️ Refer to your machine manual!

Switching on the machine and traversing the reference points can vary depending on the machine tool.

If your machine is equipped with absolute encoders, you can leave out crossing the reference points.

ℹ️ If you intend only to edit or graphically simulate NC programs, you can select the **Programming** or **Test Run** operating mode immediately after switching on the control voltage, without needing to reference the axes.

You can neither set a preset nor modify a preset via the preset table without having referenced the axes. The control issues the **Traverse reference points** message.

You can traverse the reference points later. For this purpose, in **Manual operation** mode press the **PASS OVER REFERENCE** soft key.

Cross the reference points manually in the displayed sequence:

- ▶ For each axis press the **NC START** button, or
- ＞ The control is now ready for operation in the **Manual operation** mode.

As an alternative you can cross the reference points in any sequence:

- ▶ Press and hold the axis direction button for each axis until the reference point has been traversed
- ＞ The control is now ready for operation in the **Manual operation** mode.

Crossing the reference point in a tilted working plane

If the **Tilt working plane** function was active before the control was switched off, then the control automatically activates the function after restarting. This means that movements via the axis keys take place in the tilted working plane.

Before traversing the reference points you must deactivate the **Tilt the working plane** function; otherwise, the control interrupts the process with a warning. You can also reference axes not activated in the current kinematic model without needing to deactivate **Tilt the working plane**, such as a tool magazine.

Further information: "Activating manual tilting:", Page 244

> ### *NOTICE*
>
> **Danger of collision!**
>
> The control does not automatically check whether collisions can occur between the tool and the workpiece. Incorrect pre-positioning or insufficient spacing between components can lead to a risk of collision when referencing the axes.
>
> ▶ Pay attention to the information on the screen
> ▶ If necessary, move to a safe position before referencing the axes
> ▶ Watch out for possible collisions

ⓘ If the machine does not have any absolute encoders, the position of the rotary axes must be confirmed. The position shown in the pop-up window is the last position before the control was switched off.

Switch-off

> ⚙ Refer to your machine manual!
> Deactivation is a machine-dependent function.

To prevent data from being lost on switch-off, you need to shut down the operating system of the control as follows:

- ▶ Operating mode: Press the **Manual operation** key

- ▶ Press the **OFF** soft key

- ▶ Confirm with the **SHUT DOWN** soft key
- ▶ When the control displays the message **Now you can switch off** in a pop-up window, you may switch off the power supply to the control

---

NOTICE

Caution: Data may be lost!

The control must be shut down so that running processes can be concluded and data can be saved. Immediate switch-off of the control by turning off the main switch can lead to data loss not matter what state the control was in!

- ▶ Always shut down the control
- ▶ Only operate the main switch after being prompted on the screen

---

5.2 Moving the machine axes

Note

⚙️ Refer to your machine manual!
Movement of the axes via the axis direction keys can vary depending on the machine.

Moving the axis with the axis direction keys

🖐️ ▶ Operating mode: Press the **Manual operation** key

X+ ▶ Press the axis direction key and hold it down as long as you wish the axis to move; or

X+ ▶ To move the axis continuously: Press and hold the axis direction button and press the **NC Start** key

🔲

🔲 ▶ To stop: Press the **NC Stop** key

You can move several axes at a time with these two methods. The control then shows the feed rate. You can change the feed rate at which the axes are moved with the **F** soft key.

Further information: "Spindle speed S, feed rate F and miscellaneous function M", Page 186

If a moving task is active on the machine, the control displays the **control in operation** symbol.

Incremental jog positioning

With incremental jog positioning you can move a machine axis by a preset distance.

▶ Operating mode: Press the **Manual operation** or **Electronic handwheel** key

▶ Shift the soft-key row

▶ Select incremental jog positioning: Switch the **INCREMENT** soft key to **ON**

▶ Enter the infeed of the **linear axes** and confirm with the **CONFIRM VALUE** soft key

▶ Alternatively, confirm with the **ENT** key

▶ Use the arrow keys to position the cursor on the **rotary axis**

▶ Enter the infeed of the **rotary axes** and confirm with the **CONFIRM VALUE** soft key

▶ Alternatively, confirm with the **ENT** key

▶ Confirm with the **OK** soft key

> The increment is active.
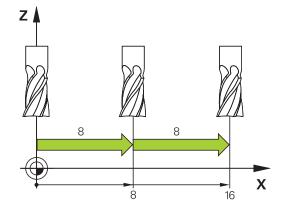
▶ Deactivate incremental jog positioning: Switch the **INCREMENT** soft key to **OFF**

ℹ If you are in the **Jog increment** menu, you can switch off incremental jog positioning with the **SWITCH OFF** soft key.

The input range for the infeed is from 0.001 mm to 10 mm.

Traverse with electronic handwheels

⚠ DANGER

Caution: Danger for the operator!

Unsecured connections, defective cables, and improper use are always sources of electrical dangers. The hazard starts when the machine is powered up!

▶ Devices should be connected or removed only by authorized service technicians

▶ Only switch on the machine via a connected handwheel or a secured connection

---

The control supports traversing with the following new electronic handwheels:

■ HR 510: Simple handwheel without display, data transfer via cable

■ HR 520: Handwheel with display, data transfer via cable

■ HR 550 FS: Handwheel with display, data transmission via radio

In addition to this, the control continues to support the cable handwheels HR 410 (without display) and HR 420 (with display).

⚙ Refer to your machine manual!

Your machine tool builder can make additional functions of the HR 5xx handwheels available.

ℹ If you wish to use the function **Handwheel superimp.** in virtual tool axis **VT**, an HR 5xx handwheel is recommended.

Further information: "Virtual tool axis (VT)(Option 44)", Page 316

The HR 520 and HR 550 FS portable handwheels are equipped with a display that is used by the control to show different types of information. In addition, you can use the handwheel soft keys for important setup functions, e.g. presetting or entering and running M functions.

As soon as you have activated the handwheel with the handwheel activation key, the operating panel is locked. The control shows this status in a pop-up window on the screen.

If several handwheels are connected to a control the handwheel key is not available on the operating panel. Activate or deactivate the handwheel via the handwheel key on the handwheel. An active handwheel must be deactivated before another handwheel can be selected.
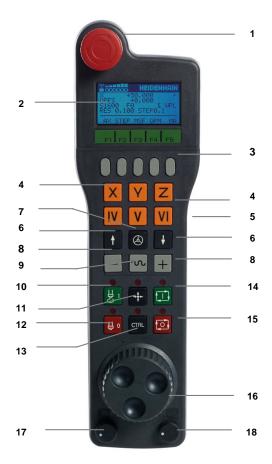
> ⚙ Refer to your machine manual!
>
> This function must be enabled and adapted by the machine tool builder.

1 EMERGENCY STOP key

2 Handwheel display for status and for selecting functions

3 Soft keys

4 Axis keys; can be exchanged by the machine manufacturer depending on the axis configuration

5 Permissive key

6 Arrow keys for defining handwheel sensitivity

7 Handwheel activation key

8 Key for traverse direction of the selected axis

9 Rapid traverse superimposing for the axis direction key

10 Spindle switch-on (machine-dependent function, key can be exchanged by the machine manufacturer)

11 **Generate NC block** key (machine-dependent function, key can be exchanged by the machine manufacturer)

12 Spindle switch-off (machine-dependent function, key can be exchanged by the machine manufacturer)

13 **CTRL** key for special functions (machine-specific function, key can be exchanged by the machine manufacturer)

14 **NC START** key (machine-dependent function, key can be exchanged by the machine manufacturer)

15 **NC STOP** key (machine-dependent function, key can be exchanged by the machine manufacturer)

16 Handwheel

17 Spindle speed potentiometer

18 Feed rate potentiometer

19 Cable connection, omitted with HR 550FS radio handwheel

Handwheel display

1 **Only with HR 550 FS radio handwheel**: Shows whether the handwheel is in the docking station or whether radio operation is active

2 **Only with HR 550 FS** handwheel: Display of field strength, six bars = maximum field strength

3 **Only with HR 550 FS radio handwheel**: Charging state of battery, six bars = maximum charging state. A bar moves from the left to the right during recharging

4 **X+50.000**: Position of the selected axis

5 *****: STIB (control in operation); program run has been started or axis is in motion

6 **OFFS +0.000**: Offset values from M118 or from the Global Program Settings (Option 44)

7 **S1000**: Current spindle speed

8 **F0:** Feed rate at which the selected axis is moving

9 **E**: Error message

If an error message appears on the control, the handwheel display shows the message **ERROR** for three seconds. Then the letter **E** is shown in the display as long as the error is pending on the control.

10 **WPL**: 3-D-ROT function is active

Depending on the settings in the 3-D ROT menu, the following is displayed:

- **VT**: The function for traversing in the tool axis direction is active
- **WP**: Basic rotation function is active

11 **RES 0,100**: Active handwheel resolution. Path traversed by the selected axis with a handwheel revolution

12 **STEP ON** or **OFF**: Incremental jog active or inactive. If the function is active, the control additionally displays the current traversing step

13 Soft-key row: Selection of various functions, described in the following sections

Special features of the HR 550 FS radio handwheel

> ⚠ **DANGER**
>
> **Caution: Danger for the operator!**
>
> Wireless handwheels, due to their rechargeable batteries and the influence of other wireless devices, are more susceptible to interference than cable-bound connections are. Ignoring the requirements for and information about safe operation leads to endangerment of the user, for example during installation or maintenance work.
>
> ▶ Check the radio connection of the handwheel for possible overlapping with other wireless devices
>
> ▶ Switch off the wireless handwheel and the handwheel holder after an operating time of 120 hours at the latest so that the control can run a functional test when it is restarted
>
> ▶ If more than one wireless handwheel is being used in a workshop, then ensure an unambiguous assignment between the handwheels and the handwheel holders (such as with color-coded stickers)
>
> ▶ If more than one wireless handwheel is being used in a workshop, then ensure an unambiguous assignment between the handwheels and the respective machine (such as with a functional test)

The HR 550 FS radio handwheel features a rechargeable battery. The battery starts charging when you put the handwheel into the holder.

The HRA 551 FS handwheel holder and the HR 550 FS handwheel together form one functional unit.

The HR 550 FS can be operated by battery for up to eight hours, before it needs recharging. When the handwheel is completely discharged, it takes about 3 hours until it is fully recharged in the handwheel holder. If you do not use the HR 550 FS, always place it into the handwheel holder intended for this. This way you can ensure that the handwheel batteries are always ready for use thanks to the contact strip on the rear side of the radio handwheel and the recharge control, and that there is a direct contact connection for the emergency stop circuit.

As soon as the handwheel is in its holder, it switches internally to cable operation. This means you can still use it even if the handwheel is fully discharged. The functions are the same as with wireless operation.

> ℹ Clean the contacts **1** in the handwheel holder and of the handwheel regularly to ensure their proper functioning.

The transmission range is amply dimensioned. If you happen to get close to the limit of the transmission range, for example in very large machines, the HR 550 FS sets off a vibrating alarm that is reliably noticed. If this happens you must reduce the distance to the handwheel holder in which the radio receiver is integrated.

NOTICE

Caution: Danger to the tool and workpiece!

The wireless handwheel triggers an emergency stop reaction if the radio transmission is interrupted, the battery is fully empty, or if there is a defect. Emergency stop reactions during machining can cause damage to the tool or workpiece.

▶ Place the handwheel in the handwheel holder when it is not in use
▶ Keep the distance between the handwheel and the handwheel holder small (pay attention to the vibration alarm)
▶ Test the handwheel before machining

If the control has triggered an emergency stop you must reactivate the handwheel. Proceed as follows:

▶ Press the **MOD** key to select the MOD function
▶ Select **Machine settings**

> SET UP
> WIRELESS
> HANDWHEEL

▶ Press the **SET UP WIRELESS HANDWHEEL** soft key
▶ Click the **Start handwheel** button to reactivate the radio handwheel
▶ To save the configuration and exit the configuration menu, press **END**

The **MOD** operating mode includes a function for commissioning and configuring the handwheel.

Further information: "Configuring the HR 550 FS radio handwheel", Page 442

Selecting the axis to be moved

You can activate the principal axes X, Y, Z and three other axes defined by the machine manufacturer directly through the axis keys. Your machine tool builder can also place the virtual axis VT directly on one of the free axis keys. If the virtual axis VT is not on one of the axis keys, proceed as follows:

▶ Press the handwheel soft key **F1** (**AX**)

> The control shows all active axes on the handwheel display. The currently active axis flashes.

▶ Select the desired axis with the handwheel soft keys **F1** (**->**) or **F2** (**<-**) and confirm with the **F3** (**OK**) handwheel soft key

⚙ Refer to your machine manual!
The machine manufacturer can also configure the turning spindle (option number 50) as an optional axis.

Setting the handwheel sensitivity

The handwheel sensitivity determines which path an axis takes per revolution of the handwheel. The handwheel sensitivity results from the defined handwheel speed of the axis and the speed level used internally by the control. The speed level describes a percentage of the handwheel speed. The control calculates a specific handwheel sensitivity value for each speed level. The resulting handwheel sensitivity values are directly selectable with the handwheel arrow keys (only if incremental jog is not active).

Taking the example of a defined handwheel speed of 1, the following handwheel sensitivity values result from the speed levels for the respective units of measure:

Resulting handwheel sensitivity levels in mm/revolution and degrees/revolution:
0.0001/0.0002/0.0005/0.001/0.002/0.005/0.01/0.02/0.05/0.1/0.2/0.5/1

Resulting handwheel sensitivity levels in in/revolution:
0.000127/0.000254/0.000508/0.00127/0.00254/0.00508/0.0127/0.0254/0.0508/0.127/0.254/0.508

Examples for resulting handwheel sensitivity values:

Defined handwheel speed	Speed level	Resulting handwheel sensitivity
10	0.01 %	0.001 mm/revolution
10	0.01 %	0.001 degrees/revolution
10	0.0127 %	0.00005 in/revolution

Moving the axes

▶ To activate the handwheel, press the handwheel button on the HR 5xx:

> Now you can operate the control only via the HR 5xx. The control displays a pop-up window with this information on the screen.

▶ Select the desired operating mode with the **OPM** soft key if necessary

▶ If required, press and hold the permissive button

▶ Use the handwheel to select the axis to be moved. Select the additional axes with the soft keys as required

▶ Move the active axis in the positive direction with the **+** key, or

▶ Move the active axis in the negative direction with the **-** key

▶ To deactivate the handwheel, press the handwheel key on the HR 5xx

> Now you can operate the control via the operating panel again.

Potentiometer settings

<div style="border:1px solid;">

⚠ DANGER

Caution: Danger for the operator!

Activating the handwheel does not automatically activate the potentiometers of the handwheel; rather the potentiometers on the operating panel of the control remain active. After an NC start on the handwheel, the control immediately begins with machining or with axis positioning, even though the potentiometers on the handwheel are set to 0 %. There is a risk of death to anybody inside the working space!

▶ Before using the handwheel, set the potentiometers of the operating panel to 0 %
▶ When using the handwheel, always also activate the potentiometers of the handwheel

</div>

The potentiometers of the machine operating panel continue to be active after you have activated the handwheel. If you want to use the potentiometers on the handwheel, proceed as follows:

▶ Press the **CTRL** and **handwheel** keys on the HR 5xx at the same time
> The control shows the soft-key menu for selecting the potentiometers in the handwheel's display.
▶ Press the **HW** soft key to activate the handwheel potentiometers

If you have activated the potentiometers on the handwheel, you must reactivate the potentiometers of the machine operating panel before deselecting the handwheel. Proceed as follows:

▶ Press the **CTRL** and **handwheel** keys on the HR 5xx at the same time
> The control shows the soft-key menu for selecting the potentiometers in the handwheel's display.
▶ Press the **KBD** soft key to activate the potentiometers of the machine operating panel

The control issues a warning if the handwheel potentiometers are still active after the handwheel has been deactivated.

Incremental jog positioning

With incremental jog positioning the control moves the currently active handwheel axis by a preset increment defined by you:

▶ Press the handwheel soft key F2 (**STEP**)

▶ Activate incremental jog positioning: Press handwheel soft key 3 (**ON**)

▶ Select the desired jog increment by pressing the **F1** or **F2** key. The smallest possible increment is 0.0001 mm (0.00001 inches). The largest possible increment is 10 mm (0.3937 inches).

▶ Confirm the selected jog increment with soft key 4 **(OK)**

▶ With the **+** or **-** handwheel key, move the active handwheel axis in the corresponding direction

ℹ If you press and hold the **F1** or **F2** key, each time it reaches a decimal value 0 the control increases the counting increment by a factor of 10.

By also pressing the **CTRL** key, you can increase the counting increment by a factor of 100 when pressing **F1** or **F2**.

Inputting miscellaneous functions M

▶ Press the **F3** (**MSF**) handwheel soft key

▶ Press the **F1** (**M**) handwheel soft key

▶ Select the desired M function number by pressing the **F1** or **F2** key

▶ Execute the M miscellaneous function with the **NC Start** key

Entering the spindle speed S

▶ Press the **F3** (**MSF**) handwheel soft key

▶ Press the **F2** (**S**) handwheel soft key

▶ Select the desired speed by pressing the **F1** or **F2** key

▶ Activate the new speed S with the **NC Start** key

ℹ If you press and hold the **F1** or **F2** key, each time it reaches a decimal value 0 the control increases the counting increment by a factor of 10.

By also pressing the **CTRL** key, you can increase the counting increment by a factor of 100 when pressing **F1** or **F2**.

Entering the feed rate F

▶ Press the **F3** (**MSF**) handwheel soft key

▶ Press the **F3** (**F**) handwheel soft key

▶ Select the desired feed rate by pressing the **F1** or **F2** key

▶ Confirm the new feed rate F with the **F3** (**OK**) handwheel soft key

> ℹ️ If you press and hold the **F1** or **F2** key, each time it reaches a decimal value 0 the control increases the counting increment by a factor of 10.
>
> By also pressing the **CTRL** key, you can increase the counting increment by a factor of 100 when pressing **F1** or **F2**.

Presetting

> ⚙️ Refer to your machine manual!
>
> The machine tool builder can disable presetting in individual axes.

▶ Press the **F3** (**MSF**) handwheel soft key

▶ Press the **F4** (**PRS**) handwheel soft key

▶ If required, select the axis in which the preset is to be set.

▶ Zero the axis with the handwheel soft key **F3** (**OK**) or with **F1** and **F2** set the desired value and then confirm with **F3** (**OK**). By also pressing the **CTRL** key, you can increase the counting increment to 10

Changing modes of operation

With the handwheel soft key **F4** (**OPM**) you can use the handwheel to switch the operating mode, provided that the current status of the control allows a mode change.

▶ Press the **F4** (**OPM**) handwheel soft key

▶ Select the desired operating mode by handwheel soft key

- MAN: **Manual operation**
  MDI: **Positioning w/ Manual Data Input**
  SGL: **Program run, single block**
  RUN: **Program run, full sequence**

Generating a complete traversing block

> ⚙️ Refer to your machine manual!
>
> Your machine tool builder can assign any function to the **Generate NC block** handwheel key.

▶ Select **Positioning w/ Manual Data Input** operating mode

▶ If required, use the arrow keys on the control's keyboard to select the NC block after which the new traversing block is to be inserted

▶ Activate the handwheel

▶ Press the **Generate NC block** key on the handwheel

> The control inserts a complete traversing block containing all axis positions selected through the MOD function.

Features in the program run modes of operation

You can use the following functions in the Program Run modes of operation:

- The **NC Start** key (**NC Start** handwheel key)
- The **NC Stop** key (**NC Stop** handwheel key)
- After the **NC Stop** key has been pressed: Internal stop (handwheel soft keys **MOP** and then **Stop**)
- After the **NC STOP** key has been pressed: Traverse manual axes (handwheel soft keys **MOP** and then **MAN**)
- Returning to the contour after the axes were moved manually during a program interruption (**MOP** and then **REPO** handwheel soft keys). The handwheel soft keys, which function similarly to the screen soft keys, are used for operating.
  Further information: "Returning to the contour", Page 297
- On/off switch for the Tilt working plane function (handwheel soft keys **MOP** and then **3D**)

5.3 Spindle speed S, feed rate F and miscellaneous function M

Application

In the **Manual operation** and **Electronic handwheel** operating modes, you can enter the spindle speed S, feed rate F and the miscellaneous functions M with soft keys.

Further information: "Entering miscellaneous functions M and STOP ", Page 310

⚙	Refer to your machine manual!
	The machine tool builder defines which miscellaneous functions are available on the machine and which are allowed in the **Manual operation** mode.

Entering values

Spindle speed S, miscellaneous function M

Proceed as follows to enter the spindle speed:

S	▶ Press the **S** soft key
	> The control shows the **Spindle speed S =** dialog in a pop-up window.
	▶ Enter **1000** (spindle speed)
⏹	▶ Press the **NC start** key to load the value

The spindle speed with the entered speed **S** is started with a miscellaneous function **M**. Input a miscellaneous function **M** in the same way.

The control shows the current spindle speed in the status display. If the spindle speed is less than 1000, the control also shows a decimal place that has been entered.

Feed rate F

Proceed as follows to enter the feed rate:

| F |

▶ Press the **F** soft key
> The control opens a pop-up window.
▶ Enter a feed rate

| ENT |

▶ Confirm with the **ENT** key

The following is valid for feed rate F:

- If you enter F=0, then the feed rate that the machine tool builder has defined as minimum feed rate is effective
- If the feed rate entered exceeds the maximum value that has been defined by the machine tool builder, then the value defined by the machine tool builder is effective
- F is not lost during a power interruption
- The control displays the feed rate.
  - When **3D ROT** is active the machining feed rate is shown if several axes are moved
  - If **3-D ROT** is inactive, the feed rate display remains empty when more than one axis is moved simultaneously

The control shows the current feed rate in the status display.

- If the feed rate is less than 10, the control also shows one of the decimal place that have been entered.
- The control displays two decimal places if the feed rate is less than 1.

Changing the spindle speed and feed rate

With the potentiometers you can vary the spindle speed **S** and feed rate **F** from 0 % to 150 % of the set value.

The feed rate potentiometer only lowers the programmed feed rate, not the feed rate calculated by the control.

⚙ The override for spindle speed is only functional on machines with infinitely variable spindle drive.

Feed rate limit F MAX

⚙ Refer to your machine manual!
The feed-rate limit depends on the machine.

The **F MAX** soft key enables you to reduce the feed rate for all operating modes. The reduction applies to all rapid traverse and feed rate movements. The value you enter remains active after switch-off or switch-on.

The **F MAX** soft key is available in the following operating modes:

- **Program run, single block**
- **Program run, full sequence**
- **Positioning w/ Manual Data Input**

Procedure

To activate the feed-rate limit F MAX, proceed as follows:

▶ Operating mode: Press the
Positioning w/ Manual Data Input key

▶ Press the **F MAX** soft key

▶ Enter the desired maximum feed rate

▶ Press the OK soft key

5.4 Optional safety concept (functional safety FS)

Miscellaneous

> ⚙ Refer to your machine manual!
> You machine tool builder adapts the HEIDENHAIN safety system to your machine.

Every machine tool operator is exposed to certain risks. Although protective devices can prevent access to dangerous points, the operator must also be able to work on the machine without this protection (e.g. protective door opened). Several guidelines and regulations to minimize these risks have been developed within the last few years.

The integrated safety design from HEIDENHAIN complies with **Performance-Level d** as per EN 13849-1 and **SIL 2** as per IEC 61508. The safety-related operating modes correspond to EN 12417 and assure extensive operator protection.

The basis of the HEIDENHAIN safety concept is the dual-channel processor structure, which consists of the main computer (MC) and one or more drive controller modules (CC= control computing unit). All monitoring mechanisms are designed redundantly in the control systems. Safety-relevant system data are subject to a mutual cyclic data comparison. Safety-relevant errors always lead to safe stopping of all drives through defined stop reactions.

Defined safety functions are triggered and safe operating statuses are achieved via safety-relevant inputs and outputs (dual-channel implementation), which have an influence on the system in all operating modes.

In this chapter you will find explanations of the functions that are additionally available on a control with functional safety.

Explanation of terms

Safety-related operating modes

Description	Brief description
SOM_1	Safe operating mode 1: Automatic operation, production mode
SOM_2	Safe operating mode 2: Set-up mode
SOM_3	Safe operating mode 3: Manual intervention; only for qualified operators
SOM_4	Safe operating mode 4: Advanced manual intervention, process monitoring

Safety functions

Description	Brief description
SS0, SS1, SS1F, SS2	Safe stop: safe stopping of all drives using different methods
STO	Safe torque off: Energy supply to the motor is interrupted. Provides protection against unexpected start of the drives
SOS	Safe operating stop. Provides protection against unexpected start of the drives
SLS	Safely-limited speed. Prevents the drives from exceeding the specified speed limits when the protective door is opened

Additional status displays

On a control with functional safety FS, the general status display contains additional information about the current status of safety functions. The control shows this information as operating states of the status displays **T**, **S**, and **F**.

Status display	Brief description
STO	Energy supply to the spindle or a feed drive is interrupted.
SLS	Safely limited speed: A safely limited speed is active
SOS	Safe operating stop: Safe operating stop is active.
STO	Safe torque off: Energy supply to the motor is interrupted.

The control displays an icon to show the status of the axes:

Button	Short description
✔	The axis has been tested or does not need to be tested.
⚠	The axis has not been tested, but must be tested to ensure safe operation. **Further information:** "Checking the axis positions", Page 192
⚠	The axis is not monitored by functional safety or is not configured as a safe axis.

The control shows the active safety-related mode of operation with an icon in the header to the right of the operating mode text:

Icon	Safety-related operating mode
SOM 1	**SOM_1** operating mode active
SOM 2	**SOM_2** operating mode active
SOM 3	**SOM_3** operating mode active
SOM 4	**SOM_4** operating mode active

Checking the axis positions

⚙ Refer to your machine manual!
This function must be adapted by your machine manufacturer.

After switch-on, the control checks whether the position of an axis matches the position directly after switch-off. If a deviation occurs or functional safety detects that a change has been made, this axis is marked on the position display. A red warning triangle is also shown on the status display. Axes that are marked can no longer be moved while the door is opened. In such cases you must approach a test position for the axes in question.

Proceed as follows:

▶ Select **Manual operation** mode
▶ Press the **MOVE TO INSPECTION POSITION** soft key
> The control displays the axes that have not been tested.
▶ Press the **SELECT AXIS** soft key
▶ Select the desired axis by soft key, if required.
▶ Alternative: Press the **POSIT. LOGIC** soft key
▶ Press the **NC start** key
> The axis moves to the test position.
> After the test position has been reached, a message appears.
▶ Press the **permissive button** on the machine operating panel
> The control displays the axis as a tested axis.
▶ Repeat this procedure for all axes that you want to move to the test position

---

NOTICE

Danger of collision!

The control does not automatically check whether collisions can occur between the tool and the workpiece. Incorrect pre-positioning or insufficient spacing between components can lead to a risk of collision while approaching the test positions.

▶ If necessary, move to a safe position before approaching the test positions
▶ Watch out for possible collisions

⚙ Refer to your machine manual!
The location of the test position is specified by your machine tool builder.

Activating feed-rate limitation

> ⚙ Refer to your machine manual!
> This function must be adapted by your machine manufacturer.

With this function you can prevent the SS1 reaction (safe stopping of drives) from being triggered when the protective door is opened.

If you press the **F LIMITED** soft key, the control will limit the speed of the axes and of the spindle(s) to the values defined by the machine tool builder. The limitation depends on the safe SOM_x operating mode selected with the aid of the keylock switch. If SOM_1 is active, the axes and spindles are brought to a stop, because only then will you be allowed to open the guard doors in SOM_1.

> ▶ Select **Manual operation** mode

> ▶ Shift the soft-key row

> ▶ Switch on/off feed rate limit

5.5 Preset management

Note

i	It is essential that you use the preset table in the following cases:
	■ If your machine is equipped with rotary axes (tilting table or swivel head) and you work with the **Tilt working plane** function
	■ If your machine is equipped with a spindle-head changing system
	■ If up to now you have been working with older controls with REF-based datum tables
	■ You wish to machine several identical workpieces that are aligned differently

The preset table can contain any number of rows (presets). To optimize the file size and the processing speed, only use as many rows as you need to manage your presets.

For safety reasons, new rows can be inserted only at the end of the preset table.
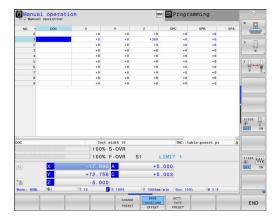
⚙	Refer to your machine manual!
	The machine tool builder specifies the default values to be used for each column of a new line.

Presets and pallet reference points

If you work with pallets, please note that the presets stored in the preset table are relative to an activated pallet reference point.

Further information: "Pallets", Page 365

Creating and activating an INCH-based preset table

ℹ️ If you choose to switch the units for your control to **INCH**, the preset table unit of measure will not be adjusted automatically.

If you want to change the unit of measure in the table as well, you need to create a new preset table.

To create and activate an **INCH**-based preset table, proceed as follows:

⇨ ▶ Select the **Programming** operating mode

PGM MGT
▶ Open the file manager
▶ Open the **TNC:\table** folder
▶ Rename the **preset.pr** file, e.g. to **preset_mm.pr**
▶ Create a new file and name it **preset_inch.pr**

INCH
▶ Select the **INCH** unit of measure
> The control opens the new, empty preset table.
> The control displays an error message related to a missing prototype file.
▶ Delete the error message

APPEND N LINES AT END
▶ Append lines, e.g. 10 lines
> The control adds the lines
▶ Position the cursor in the **ACTNO** column of line **0**
▶ Enter **1**

ENT
▶ Confirm your input

PGM MGT
▶ Open the file manager
▶ Rename the **preset_inch.pr** file to **preset.pr**

🖐 ▶ Select **Manual operation** mode

PRESET MANAGEMENT
▶ Open Preset management
▶ Review the preset table

ℹ️ The tool table is another table where the unit of measure is not adjusted automatically.

Further information: "Creating and activating an INCH-based tool table", Page 136

Saving presets in the table

⚙️ Refer to your machine manual!

The machine tool builder can disable presetting in individual axes.

The machine tool builder can define a different path for the preset table.

The preset table has the name **PRESET.PR**, and is saved in the **TNC:\table** directory by default.

PRESET.PR is editable in the **Manual operation** and **Electronic handwheel** operating modes only after you have pressed the **CHANGE PRESET** soft key. You can open the **PRESET.PR** preset table in the **Programming** operating mode but not edit it.

There are several methods for saving presets and basic rotations in the preset table:

- Manual input
- Using the touch probe cycles in the **Manual operation** and **Electronic handwheel** operating modes
- Using touch probe cycles 400 to 402 and 410 to 419 in automatic mode
  Further information: Cycle Programming User's Manual

ℹ️ Operating notes:

- In the 3-D ROT menu, you can specify that the basic rotation is also effective in the **Manual Operation** mode.
  Further information: "Activating manual tilting:", Page 244
- When presetting, the positions of the tilting axes must match the tilted situation.
- The behavior of the control during presetting depends on the setting in the optional machine parameter **chkTiltingAxes** (no. 204601).
  Further information: "Introduction", Page 206
- **PLANE RESET** does not reset the active 3-D ROT.
- In row 0 the control always saves the preset that you most recently set manually via the axis keys or via soft key. If the preset set manually is active, the control displays the text **PR MAN(0)** in the status display.

Copying the preset table

Copying the preset table into another directory (for data backup) is permitted. Write-protected lines are also write-protected in the copied tables.

Do not change the number of lines in the copied tables! If you want to reactivate the table, this may lead to problems.

To activate the preset table copied to another directory you have to copy it back.

If you select a new preset table, you need to reactivate the preset.

Manually saving the presets in the preset table

Proceed as follows in order to save presets in the preset table:

▶ Select the **Manual operation** mode

▶ Move the tool slowly until it touches (scratches) the workpiece surface, or position the measuring dial correspondingly

▶ Press the **PRESET MANAGEMENT** soft key
> The control opens the preset table and positions the cursor in the line of the active preset.

▶ Press the **CHANGE PRESET** soft key
> The control displays all available input options in the soft-key row.

▶ Select the line in the preset table that you want to change (the line number is the preset number)

▶ If needed, select the column in the preset table that you want to change

▶ Use the soft keys to select one of the available input options

Input options

Soft key	Function
	Directly transfer the actual position of the tool (the measuring dial) as the new preset: This function only saves the preset in the axis in which the cursor is currently positioned
ENTER PRESET AGAIN	Assign any value to the actual position of the tool (the measuring dial): This function only saves the preset in the axis in which the cursor is currently positioned. Enter the desired value in the pop-up window
CORRECT THE PRESET	Incrementally shift a preset already stored in the table: This function only saves the preset in the axis in which the cursor is currently positioned. Enter the desired compensation value with the correct sign in the pop-up window. If inch display is active: Enter the value in inches, and the control will internally convert the entered value to mm
EDIT CURRENT FIELD	Directly enter the new preset without calculation of the kinematics (axis-specific). Only use this function if your machine has a rotary table, and you want to set the preset to the center of the rotary table by entering 0. This function only saves the value in the axis in which the cursor is currently positioned. Enter the desired value in the pop-up window. If inch display is active: Enter the value in inches, and the control will internally convert the entered value to mm
BASE TRANSFORM. OFFSET	Select the **BASE TRANSFORM./OFFSET** view. The standard **BASE TRANSFORM.** view shows the X, Y and Z columns. Depending on the machine, the columns SPA, SPB, and SPC are displayed in addition. The control saves the basic rotation here (with the Z tool axis the control uses the SPC column). The **OFFSET** view shows the offset values for the preset.
SAVE ACTIVE PRESET	Write the currently active preset to a selectable line in the table: This function saves the preset in all axes, and then activates the appropriate line in the table automatically. If inch display is active: Enter the value in inches, and the control will internally convert the entered values to mm

Editing the preset table

Soft key	Editing function in table mode
BEGIN	Select the table start
END	Select the table end
PAGE	Select the previous page in the table
PAGE	Select the next page in the table
CHANGE PRESET	Select the functions for entry of presets
BASE TRANSFORM. OFFSET	Choose between showing the Basic Transformation or the Axis Offset
ACTI- VATE PRESET	Activate the preset of the selected line of the preset table
APPEND N LINES AT END	Add several lines at end of table
COPY FIELD	Copy the field that is currently highlighted
PASTE FIELD	Insert the copied field
RESET LINE	Reset the selected line: The control enters **-** into all columns.
INSERT LINE	Insert a line as last line in the table
DELETE LINE	Delete the last line in the table

Protecting presets from being overwritten

You can use the **LOCKED** column to protect any lines in the preset table from being overwritten. The write-protected lines are color-highlighted in the preset table.

If you want to overwrite a write-protected line with a manual touch probe cycle, confirm with **OK** and enter the password (where password-protected).

NOTICE

Caution: Data may be lost!

Lines that were locked with the **LOCK / UNLOCK PASSWORD** function can be unlocked only with the selected password. Forgotten passwords cannot be reset. This means that locked lines would be locked permanently. The preset table would thus no longer be fully usable.

▶   Prefer the alternative function **LOCK / UNLOCK**

▶   Note down your passwords

Proceed as follows to protect a preset from being overwritten:

| CHANGE PRESET | ▶   Press the **CHANGE PRESET** soft key |

| → | ▶   Select the **LOCKED** column |

| EDIT CURRENT FIELD | ▶   Press the **EDIT CURRENT FIELD** soft key |

Protection for a preset without using a password:

| LOCK / UNLOCK | ▶   Press the **LOCK / UNLOCK** soft key |
| | >   The control writes an **L** in the **LOCKED** column. |

Use a password to protect a preset:

| LOCK / UNLOCK PASSWORD | ▶   Press the **LOCK / UNLOCK PASSWORD** soft key |

	▶ Enter the password in the pop-up window
OK	▶ Confirm with the **OK** soft key or with the **ENT** key:
	> The control writes **###** in the **LOCKED** column.

Canceling write-protection

To edit a line you have previously write-protected, proceed as follows:

CHANGE PRESET	▶ Press the **CHANGE PRESET** soft key

→	▶ Select the **LOCKED** column

EDIT CURRENT FIELD	▶ Press the **EDIT CURRENT FIELD** soft key

Preset protected without a password:

LOCK / UNLOCK	▶ Press the **LOCK / UNLOCK** soft key > The control cancels the write-protection.

Preset protected with a password:

LOCK / UNLOCK PASSWORD	▶ Press the **LOCK / UNLOCK PASSWORD** soft key

OK	▶ Enter the password in the pop-up window ▶ Confirm with the **OK** soft key or with the **ENT** key > The control cancels the write-protection.

Activating a preset

Activate the preset in the Manual operation mode

NOTICE
Caution: Significant property damage!
Undefined fields in the preset table behave differently from fields defined with the value **0**: Fields defined with the value **0** overwrite the previous value when activated, whereas with undefined fields the previous value is kept.
▶ Before activating a preset, check whether all columns contain values.

Operating notes:
- When activating a preset from the preset table, the control resets any active datum shift, mirroring, rotation, or scaling factor.
- On the other hand, the **Tilt working plane** function (Cycle **19** or **PLANE**) remains active.

▶ Select the **Manual operation** mode

▶ Press the **PRESET MANAGEMENT** soft key

▶ Select the preset number that you want to activate

▶ Or, with the **GOTO** key, select the preset number that you want to activate

▶ Confirm with the **ENT** key

▶ Press the **ACTIVATE PRESET** soft key

▶ Confirm activation of the preset
> The control sets the display and the basic rotation.

▶ Exit the preset table

Activating a preset in an NC program

Use Cycle 247 in order to activate presets from the preset table during program run. In Cycle 247 you define the number of the preset to be activated.

Further information: Cycle Programming User's Manual

5.6 Presetting without a 3-D touch probe

Note

When presetting, you set the control display to the coordinates of a known workpiece position.

i	All manual probe functions are available with a 3-D touch probe. **Further information:** "Presetting with a 3-D touch probe ", Page 232

⚙	Refer to your machine manual! The machine tool builder can disable presetting in individual axes.

Preparation

▶ Clamp and align the workpiece
▶ Insert the zero tool with known radius into the spindle
▶ Ensure that the control is showing the actual positions

Presetting with an end mill

▶ Select **Manual operation** mode

▶ Move the tool slowly until it touches (scratches) the workpiece surface
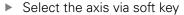
Setting a preset in an axis:

▶ Select the axis

> The control opens the **PRESETTING Z=** dialog window

▶ Alternative: Press the **SET PRESET** soft key

▶ Select the axis via soft key

▶ Zero tool in spindle axis: Set the display to a known workpiece position (here, 0) or enter the thickness d of the shim. In the tool axis: Consider the tool radius

Repeat the process for the remaining axes.
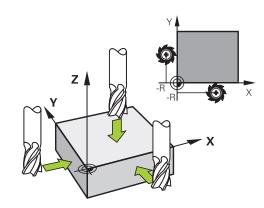
If the tool in the tool axis has already been set, set the display of the tool axis to the length L of the tool or enter the sum Z=L+d.

	Operating notes:
🅸	■ The control automatically saves the preset set with the axis keys in row 0 of the preset table. ■ If the machine tool builder has locked an axis, then you cannot set a preset in that axis. The soft key for that axis is then not visible. ■ The behavior of the control during presetting depends on the setting in the optional machine parameter **chkTiltingAxes** (no. 204601). **Further information:** "Introduction", Page 206

Using touch probe functions with mechanical probes or measuring dials

If you do not have an electronic 3-D touch probe on your machine, you can also use all the previously described manual touch probe functions (exception: calibration function) with mechanical probes or by simply touching the workpiece with the tool.

Further information: "Using a 3-D touch probe ", Page 206

In place of the electronic signal generated automatically by a 3-D touch probe during probing, you can manually initiate the trigger signal for capturing the **probing position** by pressing a key.

Proceed as follows:

▶ Select any touch probe function by soft key

▶ Move the mechanical probe to the first position to be captured by the control.

▶ To capture the position: Press the **Actual-position-capture** soft key

> The control saves the current position.

▶ Move the mechanical probe to the next position to be captured by the control.

▶ To capture the position: Press the **Actual-position-capture** soft key

> The control saves the current position.

▶ If required, move to additional positions and capture as described previously

▶ **Preset**: In the menu window, enter the coordinates of the new preset, confirm with the **SET PRESET** soft key, or write the values to a table
Further information: "Writing measured values from the touch probe cycles to a datum table", Page 215
Further information: "Writing measured values from the touch probe cycles to the preset table", Page 216

▶ Terminate the probing function: Press the **END** key

ⓘ If you try to set a preset in a locked axis, the control will issue either a warning or an error message, depending on what the machine tool builder has defined.

5.7 Using a 3-D touch probe

Introduction

The behavior of the control during presetting depends on the setting in the optional machine parameter **chkTiltingAxes** (no. 204601):

- **chkTiltingAxes: NoCheck** The control does not check whether the current coordinates of the rotary axes (actual positions) agree with the tilt angles that you defined.

- **chkTiltingAxes: CheckIfTilted** With an active tilted working plane, the control checks during presetting in the X, Y, and Z axes whether the current coordinates of the rotary axes agree with the tilt angles that you defined (3-D ROT menu). If the positions do not match, the control opens the **Working plane is inconsistent** menu.

- **chkTiltingAxes: CheckAlways** With an active tilted working plane, the control checks during presetting in the X, Y, and Z axes whether the current coordinates of the rotary axes match. If the positions do not match, the control opens the **Working plane is inconsistent** menu.

ℹ   Operating notes:

- If the checking function is off, the **PL** and **ROT** probing functions use a rotary axis position that is equal to 0 in their calculations.

- Always set a preset in all three principal axes. This clearly and correctly defines the preset. That way you also taken into account possible deviations resulting from the tilting of the axes.

- The control issues an error message if you do not use a 3-D touch probe for presetting and the positions do not match.

If the machine parameter has not been set, the control checks as if **chkTiltingAxes: CheckAlways** were set

Behavior with tilted axes

If the positions do not match, the control opens the **Working plane is inconsistent** menu.

Soft key	Function
ADOPT 3-D ROTAT. STATUS	In the 3-D ROT menu, the control sets **Manual Operation 3-D ROT** to **Active**. The linear axes move in a tilted working plane. **Manual Operation 3-D ROT** remains active until you set it to **Inactive**.
IGNORE 3-D ROTAT. STATUS	The control ignores the tilted working plane. The preset defined applies only to this tilted condition.
ALIGN ROT. AXES	The control positions the rotary axes ax specified in the 3-D ROT menu and sets **Manual Operation 3-D ROT** to **Active**. **Manual Operation 3-D ROT** remains active until you set it to **Inactive**.

Aligning the rotary axes

> ### *NOTICE*
>
> **Danger of collision!**
>
> The control does not run a collision test before aligning the rotary axes. Danger of collision if there is no pre-positioning movement.
>
> ▶ Move to a safe position before the alignment

To align the rotary axes, proceed as follows:

▶ Press the **ALIGN ROT. AXES** soft key

▶ Define the feed rate, if required
▶ Specify tilting, if required
  - **NO SYM**
  - **SYM +**
  - **SYM -**

▶ Select the positioning behavior

▶ Press the **NC start** key
> The control aligns the axes. The Tilt working plane function becomes active for this.

> ℹ Tilting can only be selected if you have set **Manual Operation 3-D ROT** to **active**.
>
> **Further information:** User's Manual Conversational Programming

Overview

The following touch probe cycles are available in the **Manual operation** mode:

> ⚙ Refer to your machine manual!
> The control must be specifically prepared by the machine tool builder for the use of a 3-D touch probe.

> ℹ HEIDENHAIN only gives warranty for the function of the probing cycles if HEIDENHAIN touch probes are used.

Soft key	Function	Page
CALIBRATE TS	Calibrating the 3-D Touch Probe	217
PROBING PL	Measuring a 3-D basic rotation by probing a plane	228
PROBING ROT	Measuring a basic rotation using a line	225
PROBING POS	Setting the preset on any axis	233
PROBING P	Setting a corner as preset	234
PROBING CC	Setting a circle center as preset	235
PROBING CL	Setting the centerline as preset	238
TCH PROBE TABLE	Touch probe system data management	See Cycle Programming User's Manual

i Operating notes:
- Touch probe functions deactivate the **Global Program Settings** temporarily.
- In turning mode you can use all manual touch probe cycles, except the **Probe corner** and **Probe plane** cycles. In turning mode, the measured values of the X axis equal diameter values.
- To use the touch probe in Turning mode, you must calibrate the touch probe separately in Turning mode. As the factory default setting of the rotary spindle may vary between Milling Mode and Turning mode, you must calibrate the touch probe without any center offset. You can create additional tool data for the touch probe, e.g. as an indexed tool.
- If the function for orienting the touch probe to the programmed probe direction is active, the number of spindle revolutions is limited when the guard door is open. In some cases, the direction of spindle rotation will change and positioning will not always follow the shortest path.

Further information: Cycle Programming User's Manual

Traverse movements with a handwheel with display

With a handwheel with display, it is possible to transfer control to the handwheel during a manual touch probe cycle.

Proceed as follows:

▶ Start the manual touch probe cycle
▶ Position the touch probe at a position near the first touch point
▶ Probe the first touch point
▶ Activate the handwheel on the handwheel
> The control shows the **Handwheel active** pop-up window.
▶ Position the touch probe at a position near the second touch point
▶ Deactivate the handwheel on the handwheel
> The control closes the pop-up window.
▶ Probe the second touch point
▶ If necessary, set the preset
▶ End the probing function

i If the handwheel is active, you cannot start the touch probe cycles.

Suppressing touch probe monitoring

Suppress touch probe monitoring

If the control does not receive a stable signal from the touch probe, the **TCH PROBE MONITOR OFF** soft key is displayed.

Proceed as follows to deactivate touch-probe monitoring:

▶ Select **Manual operation** mode

TCH PROBE
MONITOR
OFF

▶ Press the **TCH PROBE MONITOR OFF** soft key
> The control disables touch-probe monitoring for 30 seconds.
▶ If required, move the touch probe so that the control receives a stable signal from it.

As long as touch-probe monitoring is switched off, the control displays the error message **The touch probe monitor is deactivated for 30 seconds**. This error message automatically clears itself after 30 seconds.

> ⓘ   If the touch probe sends a stable signal within the 30 seconds, then touch-probe monitoring reactivates itself automatically and the error message is cleared.

NOTICE

Danger of collision!

While touch-probe monitoring is deactivated, the control will not perform collision checking. Thus, you must ensure that the touch probe can be positioned safely. There is a risk of collision if you choose the wrong direction of traverse!

▶ Carefully move the axes in the **Manual operation** mode

Functions in touch probe cycles

Soft keys that are used to select the probing direction or a probing routine are displayed in the manual touch probe cycles. The soft keys displayed vary depending on the respective cycle:

Soft key	Function
X +	Select the probing direction
╤	Capture the actual position
⊕	Probe hole (inside circle) automatically
▦	Probe stud (outside circle) automatically
PROBING CC	Probe a model circle (center point of several elements)
●	Select a paraxial probing direction for probing of holes, studs and model circles

Automatic probing routine for holes, studs and model circles

> ### *NOTICE*
>
> **Danger of collision!**
>
> The control does not perform an automatic collision check with the stylus. During automatic probing procedures the control positions the touch probe to the probing positions automatically. There is a risk of collision if pre-positioning was not correct or if obstacles have been ignored.
>
> ▶ Program a suitable pre-position
> ▶ Use safety clearances to take obstacles into account

If you use a probing routine for automatic probing of a hole, stud, or a pattern circle, the control opens a form with the required entry fields.

Input fields in the Measure stud and Measure hole forms

Input field	Function
Stud diameter? or **Hole diameter**	Diameter of probe contact (optional for holes)
Safety clearance?	Distance to the probe contact in the plane
Incr. clearance height?	Positioning of touch probe in spindle axis direction (starting from the current position)
Starting angle?	Angle for the first probing operation (0° = positive direction of principal axis, i.e. in X+ for spindle axis Z). All other probe angles result from the number of touch points.
Number of touch points?	Number of probing operations (3 to 8)
Angular length?	Probing a full circle (360°) or a circle segment (angular length<360°)

Automatic probing routine:

▶ Pre-position touch probe

▶ Select the probing function: Press the **PROBING CC** soft key

▶ Hole should be probed automatically: Press the **HOLE** soft key

▶ Select paraxial probing direction

▶ Start probing function: Press the **NC start** key
> The control carries out all pre-positioning and probing processes automatically.

The control approaches the position at the feed rate **FMAX** defined in the touch probe table. The defined probing feed rate **F** is used for the actual probing operation.

🛈 Operating and programming notes:

- Before starting an automatic probing routine, you need to preposition the touch probe near the first touch point. Offset the touch probe by approximately the safety clearance opposite to the probing direction. The safety clearance is derived from the sum of the values in the touch-probe table and in the entry form.

- For inside circles with large diameters, the control can also position the touch probe on a circular arc at the feed rate **FMAX**. This requires that you enter a safety clearance for prepositioning and the hole diameter in the input form. Position the touch probe inside the hole at a position that is offset by approximately the safety clearance from the wall. Take the starting angle of the first probing process into account in pre-positioning; for example, at a starting angle of 0° the control will first probe in the positive direction of the reference axis.

Selecting the probing cycle

▶ Select the **Manual operation** or **Electronic handwheel** mode of operation

[TOUCH PROBE]

▶ Select the probing functions: Press the **TOUCH PROBE** soft key

[PROBING POS]

▶ Select the touch probe cycle by pressing the appropriate soft key, for example **PROBING POS**

> The control displays the associated menu.

Operating notes:
- When you select a manual probing function, the control opens a form displaying all data required. The content of the forms varies depending on the respective function.
- You can also enter values in some of the fields. Use the arrow keys to switch to the desired input field. You can position the cursor only in fields that can be edited. Fields that cannot be edited are dimmed.

Recording measured values from the touch probe cycles

Refer to your machine manual!
The control must be specially prepared by the machine tool builder for use of this function.

After executing the respective touch-probe cycle, the control writes the measured values to the TCHPRMAN.html file.

If you have not defined a path in the machine parameter **FN16DefaultPath** (no. 102202), the control will store the TCHPRMAN.html file in the **TNC:** main directory.

Operating notes:
- If you run several touch probes cycles in a row, the control stores the measured values below each other.

Writing measured values from the touch probe cycles to a datum table

---

ℹ️ If you want to save measured values in the workpiece coordinate system, use the **ENTER IN DATUM TABLE** function. If you want to save measured values in the basic coordinate system, use the **ENTRY IN PRESET TABLE** function.

Further information: "Writing measured values from the touch probe cycles to the preset table", Page 216

With the **ENTER IN DATUM TABLE** soft key, the control can write the values measured during any touch probe cycle to a datum table:

▶ Select any probe function

▶ Enter the desired coordinates for the datum in the designated input boxes (depends on the touch probe cycle being run)

▶ Enter the datum number in the **Number in table?** input field

▶ Press the **ENTER IN DATUM TABLE** soft key

> The control saves the datum in the indicated datum table under the entered number.

Writing measured values from the touch probe cycles to the preset table

i If you want to save measured values in the basic coordinate system, use the **ENTRY IN PRESET TABLE** function. If you want to save measured values in the workpiece coordinate system, use the **ENTER IN DATUM TABLE** function.
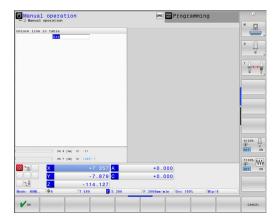
Further information: "Writing measured values from the touch probe cycles to a datum table", Page 215

With the **ENTRY IN PRESET TABLE** soft key, the control can write the values measured during any touch probe cycle to the preset table. The measured values are then stored referenced to the machine coordinate system (REF coordinates). The preset table has the name PRESET.PR, and is saved in the TNC:\table\ directory.

- ▶ Select any probe function
- ▶ Enter the desired coordinates for the preset in the designated input boxes (depends on the touch probe cycle being run)
- ▶ Enter the preset number in the **Number in table?** input field
- ▶ Press the **ENTRY IN PRESET TABLE** soft key
- > The control opens the **Overwrite active preset?** menu.
- ▶ Press the **OVERWRITE PRESET** soft key
- > The control saves the preset in the preset table under the entered number.
  - ▪ Preset number does not exist: The control does not save the line until the **CREATE LINE** soft key has been pressed (Create line in table?)
  - ▪ Preset number is protected: Press the **ENTRY IN LOCKED LINE** soft key to overwrite the active preset
  - ▪ Preset number is password-protected: Press the **ENTRY IN LOCKED LINE** soft key and enter the password to overwrite the active preset

i The control displays a note if a table row cannot be written to because of disabling. The probing function itself is not interrupted.

5.8 Calibrating 3-D touch probes

Introduction

In order to precisely specify the actual trigger point of a 3-D touch probe, you must first calibrate the touch probe, otherwise the control cannot provide precise measuring results.

ℹ Operating notes:
- Always calibrate the touch probe again in the following cases:
  - Initial configuration
  - Broken stylus
  - Stylus exchange
  - Change in the probe feed rate
  - Irregularities caused, for example, when the machine heats up
  - Change of active tool axis
- When you press the **OK** soft key after calibration, the calibration values are applied to the active touch probe. The updated tool data then become immediately effective, there is no need to retrieve the tool again.

During calibration, the control finds the effective length of the stylus and the effective radius of the ball tip. To calibrate the 3-D touch probe, clamp a ring gauge or a stud of known height and known radius to the machine table.

The control provides calibration cycles for calibrating the length and the radius:

TOUCH PROBE	▶ Press the **TOUCH PROBE** soft key
CALIBRATE TS	▶ Display the calibration cycles: Press **CALIBRATE TS**
	▶ Select the calibration cycle

Calibration cycles

Soft key	Function	Page
	Calibrating the length	218
	Measure the radius and the center offset using a calibration ring	219
	Measure the radius and the center offset using a stud or a calibration pin	219
	Measure the radius and the center offset using a calibration sphere 3-D calibrating (option 92)	219

Calibrating the effective length

ℹ️ HEIDENHAIN only gives warranty for the function of the probing cycles if HEIDENHAIN touch probes are used.
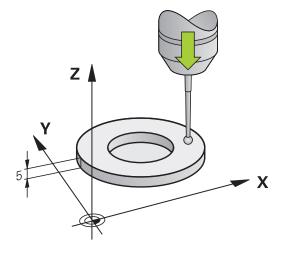
⚙️ The effective length of the touch probe is always referenced to the tool reference point. The tool reference point is often on the spindle nose (and face of the spindle). The machine manufacturer may also place the tool reference point at a different point.

▶ Set the preset in the spindle axis such that for the machine tool table Z=0.

▶ Select the calibration function for the touch probe length: Press the **CAL.** Press **L**

> The control displays the current calibration data.

▶ **Datum for length?**: Enter the height of the ring gauge in the menu window

▶ Move the touch probe to a position just above the ring gauge

▶ To change the traverse direction (if necessary), press a soft key or an arrow key

▶ Probe surface: Press **NC Start** key

▶ Check results

▶ Press the **OK** soft key for the values to take effect

▶ Press the **CANCEL** soft key to terminate the calibrating function.

> The control logs the calibration process in the TCHPRMAN.html file.

Calibrating the effective radius and compensating center misalignment

ℹ HEIDENHAIN only gives warranty for the function of the probing cycles if HEIDENHAIN touch probes are used.

When calibrating the ball-tip radius, the control executes an automatic probing routine. In the first run the control finds the midpoint of the calibration ring or stud (approximate measurement) and positions the touch probe in the center. Then, in the actual calibration process (fine measurement), the radius of the ball tip is ascertained. If the touch probe allows probing from opposite orientations, the center offset is determined during another cycle.

The property of whether or how your touch probe can be oriented is predefined for HEIDENHAIN touch probes. Other touch probes are configured by the machine tool builder.

After the touch probe is inserted, it normally needs to be aligned exactly with the spindle axis. The calibration function can determine the offset between touch probe axis and spindle axis by probing from opposite orientations (rotation by 180°) and can calculate and implement the necessary compensation.
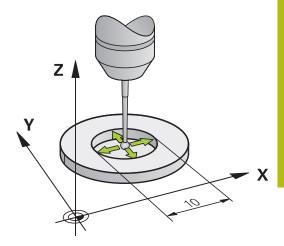
ℹ The center offset can be determined only with a suitable touch probe.

If you want to calibrate using the outside of an object, you need to preposition the touch probe above the center of the calibration sphere or calibration pin. Ensure that the touch points can be approached without collision.

The calibration routine varies depending on how your touch probe can be oriented:

- No orientation possible, or orientation in only one direction: The control executes one approximate and one fine measurement, and then ascertains the effective ball tip radius (column R in tool.t).
- Orientation possible in two directions (e.g. HEIDENHAIN touch probes with cable): The control executes one approximate and one fine measurement, rotates the touch probe by 180°, and then executes another probing routine. The center offset (CAL_OF in tchprobe.tp) is determined in addition to the radius by probing from opposite orientations.
- Any orientation possible (e.g. HEIDENHAIN touch probes with infrared transmission): The control executes one approximate and one fine measurement, rotates the touch probe by 180°, and then executes another probing routine. The center offset (CAL_OF in tchprobe.tp) is determined in addition to the radius by probing from opposite orientations.

Calibration using a calibration ring

Proceed as follows for manual calibration using a calibration ring:

▶ In the **Manual operation** mode, position the ball tip inside the bore of the ring gauge

▶ Select the calibration function: Press the **CAL. R** soft key

> The control displays the current calibration data.

▶ Enter the diameter of the ring gauge

▶ Enter the start angle

▶ Enter the number of touch points

▶ Probe: Press the **NC Start** key

> The 3-D touch probe probes all required touch points in an automatic probing routine and calculates the effective ball-tip radius. If probing from opposite orientations is possible, the control calculates the center offset.

▶ Check results

▶ Press the **OK** soft key for the values to take effect

▶ Press the **END** soft key to terminate the calibrating function.

> The control logs the calibration process in the TCHPRMAN.html file.

⚙ Refer to your machine manual!

In order to be able to determine ball-tip center misalignment, the control needs to be specially prepared by the machine manufacturer.

Calibration with a stud or calibration pin

Proceed as follows for manual calibration with a stud or calibration pin:

▶ In the **Manual operation** mode, position the ball tip above the center of the calibration pin

▶ Select the calibration function: Press the **CAL. R** soft key

▶ Enter the outside diameter of the stud

▶ Enter the safety clearance

▶ Enter the start angle

▶ Enter the number of touch points

▶ Probe: Press the **NC Start** key

> The 3-D touch probe probes all required touch points in an automatic probing routine and calculates the effective ball-tip radius. If probing from opposite orientations is possible, the control calculates the center offset.

▶ Check results

▶ Press the **OK** soft key for the values to take effect

▶ Press the **END** soft key to terminate the calibrating function.

> The control logs the calibration process in the TCHPRMAN.html file.

⚙ Refer to your machine manual!

In order to be able to determine ball-tip center misalignment, the control needs to be specially prepared by the machine manufacturer.

Calibration using a calibration sphere

Proceed as follows for manual calibration using a calibration sphere:

▶ In the **Manual operation** mode, position the ball tip above the center of the calibration sphere

▶ Select the calibration function: Press the **CAL. R** soft key

▶ Enter the outside diameter of the ball

▶ Enter the safety clearance

▶ Enter the start angle

▶ Enter the number of touch points

▶ Select Length measurement, if applicable

▶ If necessary, input the reference for the length

▶ Probe: Press the **NC Start** key

> The 3-D touch probe probes all required touch points in an automatic probing routine and calculates the effective ball-tip radius. If probing from opposite orientations is possible, the control calculates the center offset.

▶ Check results

▶ Press the **OK** soft key for the values to take effect

▶ Press the **END** soft key to terminate the calibrating function or enter the number of probing points for 3-D calibration

> The control logs the calibration process in the TCHPRMAN.html file.

⚙ Refer to your machine manual!

In order to be able to determine ball-tip center misalignment, the control needs to be specially prepared by the machine manufacturer.

3-D calibration with a calibration sphere (option 92)

In addition to calibrating with a calibration sphere, the control also enables the touch probe to be calibrated dependent on the angle. For this purpose the control probes the calibration sphere in a quarter circle in the perpendicular. The 3-D calibration data specifies the deflection behaviour of the touch probe in any probing direction.

The **3D-ToolComp** software option (option 92) is required for this.

- ▶ Execute calibration using a calibration sphere
- ▶ Enter the number of touch points
- ▶ Press the **NC Start** key
- > The 3-D touch probe probes all required touch points in an automatic probing routine.
- ▶ Press the **OK** soft key.
- ▶ Press the **END** soft key to terminate the calibrating function.
- > The control saves the deviations in a compensation value table under **TNC:\system \3D-ToolComp**.

The control creates a specific table for each calibrated touch probe. In the tool table the **DR2TABLE** column is automatically referenced to this.

Displaying calibration values

The control saves the effective length and effective radius of the touch probe in the tool table. The control saves the touch probe center offset to the touch probe table in the columns **CAL_OF1** (principal axis) and **CAL_OF2** (minor axis). You can display the values on the screen by pressing the **TCH PROBE TABLE** soft key.
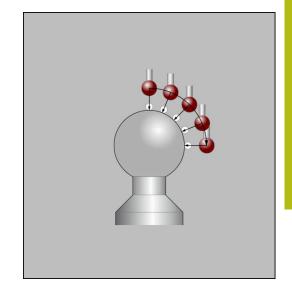
During calibration, the control automatically creates the TCHPRMAN.html log file to which the calibration values are saved.

| **i** | Ensure that the tool number in the tool table and the touch-probe number of the touch-probe table match. This is regardless of whether you want to use a touch-probe cycle in automatic mode or in **Manual operation** mode. |

| 📖 | **Further information:** Cycle Programming User's Manual |

5.9 Compensating workpiece misalignment with 3-D touch probe

Introduction

> ⚙ Refer to your machine manual!
> It depends on the machine whether you can compensate workpiece misalignment with an offset (angle for table rotation).

> ℹ HEIDENHAIN only gives warranty for the function of the probing cycles if HEIDENHAIN touch probes are used.

The control compensates workpiece misalignment either mathematically by computing a basic rotation (angle of basic rotation) or by an offset (angle for table rotation)

For this purpose, the control sets the rotation angle to the desired angle with respect to the reference axis in the working plane.

Basic rotation: The control interprets the measured angle as rotation around the tool direction, and saves the values in the columns SPA, SPB, or SPC of the preset table.
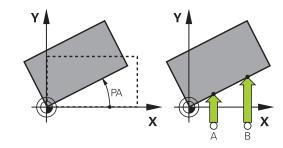
Offset: The control interprets the measured angle as a shift in each axis in the machine coordinate system, and saves the values in the columns A_OFFS, B_OFFS, or C_OFFS of the preset table.

In order to identify the basic rotation or offset, probe two points on the side of the workpiece. The sequence in which you probe the points influences the calculated angle. The measured angle goes from the first to the second probing point. You can also determine the basic rotation or offset using holes or studs.

> ℹ Operating and programming notes:
> - Select the probe direction perpendicular to the angle reference axis when measuring workpiece misalignment.
> - To ensure that the basic rotation is calculated correctly during program run, program both coordinates of the working plane in the first positioning block.
> - You can also use a basic rotation in conjunction with the **PLANE** function (except for **PLANE AXIAL**). In this case first activate the basic rotation and then the **PLANE** function.
> - You can also activate a basic rotation or offset without probing a workpiece. To do so, enter a value in the corresponding input field and press the **SET BASIC ROTATION** or **SET TABLE ROTATION** soft key.
> - The behavior of the control during presetting depends on the setting in the machine parameter **chkTiltingAxes** (no. 204601).
>   **Further information:** "Introduction", Page 206

Measuring a basic rotation

PROBING
ROT

- ▶ Press the **Probe rotation** soft key
- ＞ The control opens the **Probing of rotation** menu.
- ▶ The following input fields are displayed:
  - ■ **Angle of basic rotation**
  - ■ **Offset of rotary table**
  - ■ **Number in table?**
- ＞ The control displays any current basic rotation or offset in the input field.
- ▶ Position the touch probe at a position near the first touch point
- ▶ Select the probe direction or probing routine by soft key
- ▶ Press the **NC Start** key
- ▶ Position the touch probe at a position near the second touch point
- ▶ Press the **NC Start** key
- ＞ The control determines the basic rotation and offset and displays them.
- ▶ Press the **SET BASIC ROTATION** soft key
- ▶ Press the **END** soft key

The control logs the probing process in TCHPRMAN.html.
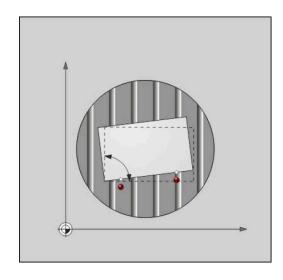
Saving the basic rotation in the preset table

- ▶ After the probing process, enter the preset number in which the control is to save the active basic rotation in the **Number in table?** input field
- ▶ Press the **BASIC ROT. IN DATUM TBL** soft key
- ＞ If appropriate, the control opens the **Overwrite active preset?** menu.
- ▶ Press the **OVERWRITE PRESET** soft key
- ＞ The control saves the basic rotation in the preset table.

Compensation of workpiece misalignment by rotating the table

There are three possibilities for compensating workpiece misalignment by rotating the table:

- Align rotary table
- Set table rotation
- Save table rotation in the preset table

Align rotary table

You can compensate the ascertained misalignment by positioning the rotary table.

🛈 Pre-position all axes before rotating the table, in order to preclude collisions resulting from compensating movements. The control additionally outputs a warning before table rotation.

▶ Press the **ALIGN ROT. TABLE** soft key after the probing procedure
> The control opens the warning.
▶ Clear with the **OK** soft key if needed
▶ Press the **NC Start** key
> The control aligns the rotary table.

Set table rotation

You can define a manual preset in the axis of the rotary table.

▶ Press the **SET TABLE ROTATION** soft key after the probing procedure
> If a basic rotation is already set, the control opens the **Reset basic rotation?** menu.
▶ Press the **DELETE BASIC ROT.** soft key
> The control deletes the basic rotation from the preset table, and inserts the offset.
▶ Or press **KEEP BASIC ROT.**
> The control inserts the offset in the preset table, and the basic rotation also remains.

Saving table rotation in the preset table

You can save the misalignment of the rotary table in any row of the preset table. The control stores the angle in the offset column of the rotary table, e.g. in the C_OFFS column for a C axis.

▶ Press the **TABLE ROT. IN DATUM TBL** soft key after the probing procedure

> If appropriate, the control opens the **Overwrite active preset?** menu.

▶ Press the **OVERWRITE PRESET** soft key

> The control saves the offset in the preset table.
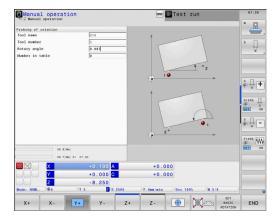
You may have to change the view in the preset table with the **BASE TRANSFORM./OFFSET** soft key for this column to be displayed.

Showing basic rotation and offset

If you select the **PROBING ROT** function, the control displays the active angle of the basic rotation in the **Angle of basic rotation** input field and the active offset in the **Offset of rotary table** input field.

In addition, the control shows the basic rotation and the offset in the **PROGRAM + STATUS** screen layout on the **STATUS POS.** tab.

When the control moves the machine axis in accordance with the basic rotation, a symbol for the basic rotation is shown in the status display.

Rescinding basic rotation or offset

▶ Select the probe function by pressing the **PROBING ROT** soft key

▶ Enter the **Angle of basic rotation**: **0**

▶ Alternative: Enter the **Offset of rotary table**: **0**

▶ Apply with the **SET BASIC ROTATION** soft key

▶ Alternative: Apply with the **SET TABLE ROTATION** soft key

▶ Terminate the probing function: Press the **END** soft key

Measuring 3-D basic rotation

You can measure the inclined position of any tilted surface by probing three positions. The **Probe in plane** function enables you to measure this inclined position and save it as a 3-D basic rotation in the preset table.

> **i** Operating and programming notes:
> - The sequence and position of the touch points determines how the control will calculate the orientation of the plane.
> - With the first two points, you specify the direction of the reference axis. Define the second point in the positive direction of the desired reference axis. The position of the third point determines the direction of the minor axis and of the tool axis. Define the third point in the positive Y axis of the desired workpiece coordinate system.
>   - 1st point: On the reference axis
>   - 2nd point: On the reference axis, in a positive direction from the first point
>   - 3rd point: On the minor axis, in a positive direction of the desired workpiece coordinate system

Optionally inputting a datum angle enables you to define the nominal direction of the probed plane.

Procedure

▶ Select the probing function: Press the **PROBING PL** soft key

> The control displays the current 3-D basic rotation.

▶ Position the touch probe at a position near the first touch point

▶ Select the probe direction or probing routine by soft key

▶ Probe: Press the **NC Start** key

▶ Position the touch probe at a position near the second touch point

▶ Probe: Press the **NC Start** key

▶ Position the touch probe near the third touch point

▶ Probe: Press the **NC Start** key.

> The control measures the 3-D basic rotation and displays the values for SPA, SPB, and SPC in relation to the active coordinate system.

▶ If required, enter the reference angle

Activate 3-D basic rotation:

▶ Press the **SET BASIC ROTATION** soft key

Save the 3-D basic rotation in the preset table:

▶ Press the **BASIC ROT. IN DATUM TBL** soft key

▶ Terminate the probing function: Press the **END** soft key

The control saves the 3-D basic rotation in the SPA, SPB, and SPC columns of the preset table.

Displaying 3-D basic rotation

If a 3-D basic rotation is saved in the active preset, the control shows the ⬚ icon for the 3-D basic rotation in the status display. The control moves the machine axes according to the 3-D basic rotation.

Aligning 3-D basic rotation

If the machine has two rotary axes and the probed 3-D basic rotation is activated, you can use the rotary axes to align the 3-D basic rotation.

NOTICE

Danger of collision!

The control does not run a collision test before aligning the rotary axes. Danger of collision if there is no pre-positioning movement.

▶ Move to a safe position before the alignment

Proceed as follows:

ALIGN ROT. AXES	▶ Press the **ALIGN ROT. AXES** soft key
	> The control displays the calculated axis angles.
	▶ Enter a feed rate
NO SYM	▶ Select a solution where necessary
	> The control activates the 3-D rotation and updates the axis-angle display.
TURN	▶ Select the positioning behavior
	▶ Press the **NC start** key
	> The control aligns the axes. The Tilt working plane function becomes active for this.

After aligning the plane, you can align the reference axis with the **Probing rot** function.

Canceling a 3-D basic rotation

PROBING PL	▶ Select the probe function by pressing the **PROBING PL** soft key
	▶ Enter 0 for all angles
	▶ Press the **SET BASIC ROTATION** soft key
	▶ Terminate the probing function: Press the **END** soft key

Comparison of offset and 3-D basic rotation

The following example shows how the two functions differ.

Offset	**3-D basic rotation**
Initial state	Initial state
Position display:	Position display:
■ Actual position	■ Actual position
■ **B** = 0	■ **B** = 0
■ **C** = 0	■ **C** = 0
Preset table:	Preset table:
■ **SPB** = 0	■ **SPB** = -30
■ **B_OFFS** = -30	■ **B_OFFS** = +0
■ **C_OFFS** = +0	■ **C_OFFS** = +0
Movement in +Z without tilting	Movement in +Z without tilting
Movement in +Z with tilting	Movement in +Z with tilting
PLANE SPATIAL with **SPA+0 SPB+0 SPC+0**	**PLANE SPATIAL** with **SPA+0 SPB+0 SPC+0**
> The orientation **is not correct**!	> The orientation is correct!
	> The next machining step will be **correct**.
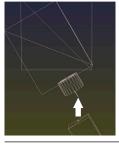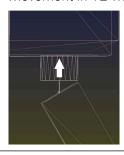
> **ℹ** HEIDENHAIN recommends to use the 3-D basic rotation because of its greater flexibility.

5.10 Presetting with a 3-D touch probe

Overview

> ⚙ Refer to your machine manual!
>
> The machine tool builder can disable presetting in individual axes.
>
> If you try to set a preset in a locked axis, the control will issue either a warning or an error message, depending on what the machine tool builder has defined.

The following soft-key functions are available for setting a preset on an aligned workpiece:

Soft key	Function	Page
PROBING POS	Setting a preset in any axis	233
PROBING P	Setting a corner as preset	234
PROBING CC	Setting a circle center as preset	235
PROBING CL	Setting the center line as preset	238

> ℹ With an active datum shift the determined value is with respect to the current preset (possibly a manual preset from the **Manual operation** mode). The datum shift is taken into account in the position display.

Presetting with active TCPM

An active TCPM is taken into account during presetting. This makes it possible to probe positions with active TCPM even if the **Tilt working plane** condition is inconsistent.

Further information: "Using a 3-D touch probe ", Page 206

> ℹ A 3-D calibration of the touch probe is necessary to obtain exact probing results.
>
> **Further information:** "Calibration using a calibration sphere", Page 222

Presetting on any axis
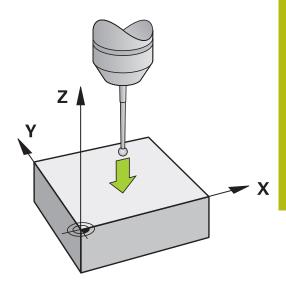
ℹ️ HEIDENHAIN only gives warranty for the function of the probing cycles if HEIDENHAIN touch probes are used.

- ▶ Select the probing function: Press the **POSITION PROBING** soft key
- ▶ Move the touch probe to a position near the touch point
- ▶ Select the axis and probing direction, e.g. Probe in direction Z-
- ▶ Probe: Press the **NC Start** key
- ▶ **Preset**: Enter the nominal coordinate
- ▶ Load with the **SET PRESET** soft key
  Further information: "Writing measured values from the touch probe cycles to a datum table", Page 215
  Further information: "Writing measured values from the touch probe cycles to the preset table", Page 216
- ▶ Terminate the probing function: Press the **END** soft key

Corner as preset

⚙	Refer to your machine manual! It depends on the machine whether you can compensate workpiece misalignment with an offset (angle for table rotation).

ℹ	HEIDENHAIN only gives warranty for the function of the probing cycles if HEIDENHAIN touch probes are used.

The "Corner as preset" probing cycle identifies the angle and intersection of two straight lines.
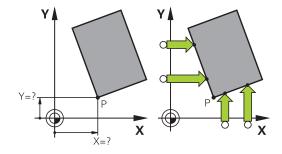
▶ Select the probing function: Press the **PROBING P** soft key

▶ Position the touch probe near the first touch point on the first workpiece edge

▶ Select the probe direction by soft key

▶ Probe: Press the **NC Start** key

▶ Position the touch probe near the second touch point on the same workpiece edge

▶ Probe: Press the **NC Start** key

▶ Position the touch probe near the first touch point on the second workpiece edge

▶ Select the probe direction by soft key

▶ Probe: Press the **NC Start** key

▶ Position the touch probe near the second touch point on the same workpiece edge

▶ Probe: Press the **NC Start** key

▶ **Preset**: Enter both coordinates of the preset in the menu window

▶ Load with the **SET PRESET** soft key
Further information: "Writing measured values from the touch probe cycles to a datum table", Page 215
Further information: "Writing measured values from the touch probe cycles to the preset table", Page 216

▶ Terminate the probing function: Press the **END** soft key

> **ℹ** You can identify the intersection of two straight lines by holes or studs and set this as the preset.

The **ROT 1** soft key activates the angle of the first straight line as basic rotation or as offset and the **ROT 2** soft key activates the angle or offset of the second straight line.

If you activate the basic rotation, the control automatically writes the positions and the basic rotation to the preset table.

If you activate the offset, the control automatically writes the positions and the offset or only the positions to the preset table.

Circle center as preset

With this function, you can set the preset at the center of bore holes, circular pockets, cylinders, studs, circular islands, etc.

Inside circle:

The control probes the inside wall of a circle in all four coordinate axis directions.

For incomplete circles (circular arcs) you can choose the appropriate probing direction.
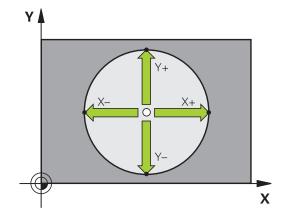
> ▶ Position the touch probe approximately in the center of the circle

PROBING CC

> ▶ Select the touch probe function: Press the **PROBING CC** soft key
> ▶ Select the soft key for the desired probing direction
> ▶ Probe: Press the **NC Start** key. The touch probe probes the inside wall of the circle in the selected direction. Repeat this process. After the third probing process, you can have the control calculate the center (four touch points are recommended)
> ▶ Terminate the probing procedure and switch to the evaluation menu: Press the **EVALUATE** soft key
> ▶ **Preset**: Enter both coordinates of the center of the circle in the menu window
> ▶ Load with the **SET PRESET** soft key
> **Further information:** "Writing measured values from the touch probe cycles to a datum table", Page 215
> **Further information:** "Writing measured values from the touch probe cycles to the preset table", Page 216
> ▶ Terminate the probing function: Press the **END** soft key

> **ℹ** The control needs at least three touch points to calculate outside or inside circles, e.g. with circle segments. More precise results are obtained with four touch points. If possible, always pre-position the touch probe to the center.

Outside circle:
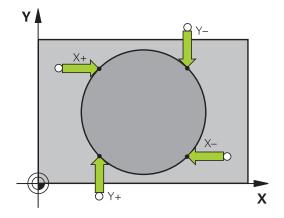
▶ Position the touch probe at a position near the first touch point outside of the circle

PROBING CC

▶ Select the touch probe function: Press the **PROBING CC** soft key

▶ Select the soft key for the desired probing direction

▶ Probe: Press the **NC Start** key. The touch probe probes the inside wall of the circle in the selected direction. Repeat this process. After the third probing process, you can have the control calculate the center (four touch points are recommended)

▶ Terminate the probing procedure and switch to the evaluation menu: Press the **EVALUATE** soft key

▶ **Preset**: Enter the coordinates of the preset

▶ Load with the **SET PRESET** soft key
Further information: "Writing measured values from the touch probe cycles to a datum table", Page 215
Further information: "Writing measured values from the touch probe cycles to the preset table", Page 216

▶ Terminate the probing function: Press the **END** soft key

Once the probing routine is completed, the control displays the current coordinates of the circle center and the circle radius.

Setting the preset using multiple holes or circular studs

The manual probing function **Probing of circular pattern** is part of the **Cir** probing function. Individual circles can be measured with paraxial probing operations.

A second soft-key row provides the **PROBING CC (probing of circular pattern)** soft key for using multiple holes or circular studs to set the preset. You can set the intersection of two or more elements as preset.

Setting the preset in the intersection of multiple holes or circular studs:

▶ Pre-position touch probe

Select the **Probing CC** probing function

▶ Select the touch probe function: Press the **PROBING CC** soft key

▶ Press the **PROBING CC (Probing of circular pattern)** soft key

Probe a circular stud

▶ Circular stud should be probed automatically: Press **Stud** soft key

▶ Enter starting angle or select using soft key

▶ Start probing function: Press the **NC start** key

Probe the hole.

▶ Hole should be probed automatically: Press the **Hole** soft key

▶ Enter starting angle or select using soft key

▶ Start probing function: Press the **NC start** key

▶ Repeat the probing procedure for the remaining elements

▶ Terminate the probing procedure and switch to the evaluation menu: Press the **EVALUATE** soft key

▶ **Preset**: Enter both coordinates of the center of the circle in the menu window

▶ Load with the **SET PRESET** soft key
   Further information: "Writing measured values from the touch probe cycles to a datum table", Page 215
   Further information: "Writing measured values from the touch probe cycles to the preset table", Page 216

▶ Terminate the probing function: Press the **END** soft key

Setting a center line as preset

▶ Select the probing function: Press the **PROBING CL** soft key

▶ Position the touch probe at a position near the first touch point

▶ Select the probing direction by soft key

▶ Probe: Press the **NC Start** key

▶ Position the touch probe at a position near the second touch point

▶ Probe: Press the **NC Start** key

▶ **Preset**: Enter the coordinates of the preset in the menu window, confirm with the **SET PRESET** soft key, or write the value to a table
Further information: "Writing measured values from the touch probe cycles to a datum table", Page 215
Further information: "Writing measured values from the touch probe cycles to the preset table", Page 216
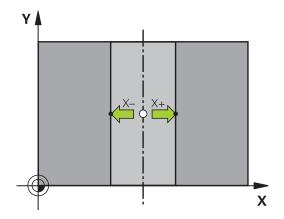
▶ Terminate the probing function: Press the **END** soft key

---
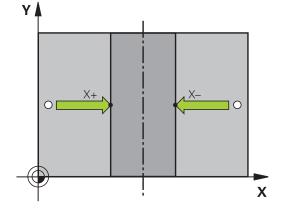
i If you desire, then after the second touch point you can change the position of the centerline in the evaluation menu, and thus the axis for setting the preset. Use the soft keys to choose between principal axis, secondary axis, and tool axis. This way you can determine the positions once, and then store them in the principal axis as well as in the secondary axis.

Measuring workpieces with a 3-D touch probe

You can also use the touch probe in the **Manual operation** and **Electronic handwheel** operating modes to perform simple measurements on the workpiece. Numerous programmable touch probe cycles are available for more complex measuring tasks.

Further information: Cycle Programming User's Manual

With a 3-D touch probe you can determine:

- Position coordinates, and from them,
- Dimensions and angles on the workpiece

Finding the coordinates of a position on an aligned workpiece

- ▶ Select the probing function: Press the **PROBING POS** soft key
- ▶ Move the touch probe to a position near the touch point
- ▶ Select the probing direction and the axis to which the coordinates relate: Use the corresponding soft keys to select
- ▶ Start the probing process: Press the **NC Start** key

The control shows the coordinates of the touch point as preset.

Finding the coordinates of a corner point on the working plane

Find the coordinates of the corner point.

Further information: "Corner as preset ", Page 234

The control displays the coordinates of the probed corner as preset.

Measuring workpiece dimensions

▶ Select the probing function: Press the **PROBING POS** soft key

▶ Position the touch probe at a position near the first touch point A

▶ Select the probing direction by soft key

▶ Probe: Press the **NC Start** key

▶ If you need the current preset later, write down the value that appears in the display

▶ Preset: Enter **0**.

▶ Cancel the dialog: Press the **END** key

▶ Select the probing function again: Press the **PROBING POS** soft key

▶ Position the touch probe at a position near the second touch point B

▶ Select the probe direction with the soft keys: Same axis but from the opposite direction

▶ Probe: Press the **NC Start** key

The **Measured value** display shows the distance between the two points on the coordinate axis.

To return to the values that were active before the length measurement:
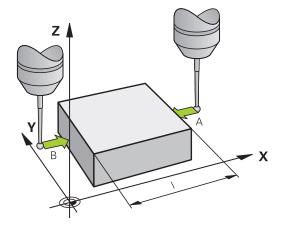
▶ Select the probing function: Press the **PROBING POS** soft key

▶ Probe the first touch point again

▶ Set the preset to the value that you wrote down previously

▶ Cancel the dialog: Press the **END** key

Measuring angles

You can use the 3-D touch probe to measure angles in the working plane. You can measure

- The angle between the angle reference axis and a workpiece edge; or

- the angle between two sides
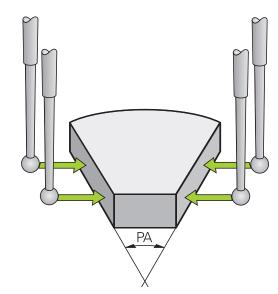
The measured angle is displayed as a value of max. 90°.

Finding the angle between the angle reference axis and a workpiece edge
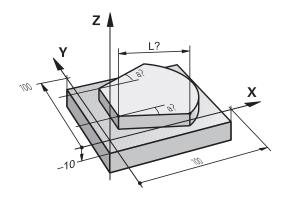
▶ Select the probe function by pressing the **PROBING ROT** soft key

▶ Rotation angle: If you wish to restore the current basic rotation later, note the value that appears under Rotation Angle

▶ Perform a basic rotation with the workpiece edge to be compared
   Further information: "Compensating workpiece misalignment with 3-D touch probe ", Page 224

▶ Press the **PROBING ROT** soft key to display the angle between the angle reference axis and the workpiece edge as the rotation angle

▶ Cancel the basic rotation, or restore the previous basic rotation

▶ Set the rotation angle to the value that you previously wrote down

Measuring the angle between two workpiece edges

▶ Select the probe function by pressing the **PROBING ROT** soft key

▶ Rotation angle: If you wish to restore the current basic rotation later, note the value that appears under Rotation Angle

▶ Perform a basic rotation with the workpiece edge to be compared
   Further information: "Compensating workpiece misalignment with 3-D touch probe ", Page 224

▶ Probe the second edge in the same way as for a basic rotation, but do not set the rotation angle to 0

▶ Press the **ROTATION PROBING** soft key to display the angle PA between the workpiece edges as the rotation angle

▶ Cancel the basic rotation, or restore the previous basic rotation by setting the rotation angle to the value that you wrote down previously

5.11 Tilting the working plane (option 8)

Application, function

> ⚙ Refer to your machine manual!
>
> The **Tilt working plane** functions are adapted to the control and the machine tool by the machine tool builder.
>
> The machine tool builder also specifies whether the programmed angles are interpreted as coordinates of the rotary axes (axis angles) or as angular components of a tilted plane (spatial angles).
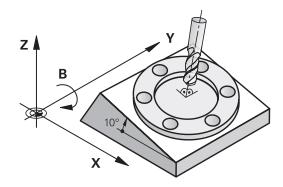
The control supports the tilting functions on machine tools with swivel heads and/or tilting tables. Typical applications are, for example, oblique holes or contours in an oblique plane. The working plane is always tilted around the active datum. The program is written as usual in a main plane, such as the X/Y plane, but is executed in a plane that is tilted relative to the main plane.

There are three functions available for tilting the working plane:

- Manual tilting with the **3-D ROT** soft key in the **Manual operation** and **Electronic handwheel** operating modes
  Further information: "Activating manual tilting:", Page 244

- Controlled tilting, Cycle **19 WORKING PLANE** in the NC program
  Further information: Cycle Programming User's Manual

- Tilting under program control, **PLANE** function in the NC program
  Further information: User's Manual for Conversational Programming or ISO Programming

The control functions for tilting the working plane are coordinate transformations. The working plane is always perpendicular to the direction of the tool axis.

When tilting the working plane, the control differentiates between two machine types:

- **Machine with tilting table**
  - You must tilt the workpiece into the desired position for machining by positioning the tilting table, for example with an L block.
  - The position of the transformed tool axis **does not change** in relation to the machine coordinate system. Thus if you rotate the table—and therefore the workpiece—by 90° for example, the coordinate system **does not rotate**. If you press the Z+ axis direction button in the **Manual operation** mode, the tool moves in Z+ direction.
  - In calculating the transformed coordinate system, the control considers only the mechanically influenced offsets of the particular tilting table (the translational components).

- **Machine with swivel head**
  - You must tilt the workpiece into the desired position for machining by positioning the swivel head, for example with an L block
  - The position of the tilted (transformed) tool axis changes in relation to the machine coordinate system. Thus if you rotate the swivel head of your machine—and therefore the tool— in the B axis by +90° for example, the coordinate system also rotates. If you press the Z+ axis direction button in the **Manual operation** mode, the tool moves in the X+ direction of the machine coordinate system.
  - In calculating the active coordinate system, the control considers both the mechanically influenced offsets of the particular swivel head (the translational components) as well as offsets caused by tilting of the tool (3-D tool length compensation).

> **i** The control only supports the **Tilt working plane** function in combination with the spindle axis Z.

Position display in a tilted system

The positions displayed in the status window **(ACTL.** and **NOML.)** are referenced to the tilted coordinate system.

In the optional machine parameter **CfgDisplayCoordSys** (no. 127501), you can specify the coordinate system in which the status display shows an active datum shift.

Limitations on working with the tilting function

- The **Actual-position capture** function is not allowed if the Tilt working plane function is active
- PLC positioning (determined by the machine tool builder) is not possible.

Activating manual tilting:

▶ Press the **3-D ROT** soft key

> The control opens the **Tilt the working plane** pop-up window.

▶ Use the arrow keys to position the cursor on the desired function

- **Manual Operation Tool Axis**
- **Manual Operation 3-D ROT**
- **Manual Operation Basic Rotation**

▶ Press the **ACTIVE** soft key

▶ If required, use the arrow keys to position the cursor on the desired rotary axis

▶ If applicable, press the **3-D ROT: AXIS ANGLE SPA. ANGLE** soft key

> The control now displays the spatial angle input fields.

▶ If required, enter the tilting angle

▶ Press the **END** key

> The input is complete.

ℹ️ By setting **Manual Operation 3-D ROT** to **Active**, you can press the **3-D ROT: AXIS ANGLE SPA. ANGLE** soft key to toggle between axis values and spatial angles.

Manual Operation Tool Axis

⚙️ Refer to your machine manual!
Your machine manufacturer enables this function.

If the Traverse in Tool Axis function is active, the control displays the 🔧 icon in the status display.

You can only move in the direction of the tool axis. The control locks all other axes.

The traverse is effective in the T-CS tool coordinate system.

Further information: "Tool coordinate system T-CS", Page 125

Manual Operation 3-D ROT
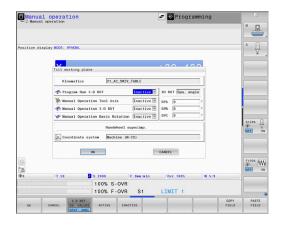
If the 3-D ROT function is active, the control displays the 🔧 icon in the status display.

All axes move in a tilted working plane.

If a basic rotation or 3-D basic rotation has additionally been saved to the preset table, it will automatically be taken into account.

The axis movements are effective in the WPL-CS working plane coordinate system.

Further information: "Working plane coordinate system WPL-CS", Page 122

Manual Operation Basic Rotation

If the basic rotation function is active, the control displays the icon in the status display.

If a basic rotation or a 3-D basic rotation has already been saved to the preset table, the control additionally displays the corresponding icon.

> **ℹ** If the **Manual Operation Basic Rotation** function is active, an active basic rotation or 3-D basic rotation is taken into account during manual movement of the axes. The control displays two icons in the status display.

The axis movements are effective in the W-CS workpiece coordinate system.

Further information: "Workpiece coordinate system W-CS", Page 120

Program Run 3-D ROT

If you set the **Tilt working plane** function to **Active** for the **Program run** mode, the tilt angle entered in the menu will be effective starting from the first NC block of the NC program to be run.

If you use Cycle **19 WORKING PLANE** or the **PLANE** function in the NC program, the angle values defined there are in effect. Angle values entered in the menu will be set to 0.

> **ℹ** The control uses the following **transformation types** for tilting:
> - **COORD ROT**
>   - if a **PLANE** function was previously executed with **COORD ROT**
>   - after **PLANE RESET**
>   - with corresponding configuration of the machine parameter **CfgRotWorkPlane** (no. 201200) by the machine tool builder
> - **TABLE ROT**
>   - if a **PLANE** function was previously executed with **TABLE ROT**
>   - with corresponding configuration of the machine parameter **CfgRotWorkPlane** (no. 201200) by the machine tool builder

> **📖** If tilting was active when the control was turned off, then the control also moves in the tilted working plane when it is turned on again.
>
> **Further information:** "Crossing the reference point in a tilted working plane", Page 171

Deactivating manual tilting

To deactivate, set the desired function to **Inactive** in the **Tilt working plane** menu.

Even if the **3D-ROT** dialog in the **Manual operation** mode is set to **Active**, resetting the tilting (**PLANE RESET**) with an active basic transformation still functions correctly.

Setting the tool-axis direction as the active machining direction

⚙ Refer to your machine manual!
Your machine manufacturer enables this function.

Using this function in the **Manual operation** and **Electronic handwheel** operating modes, you can move the tool in the direction in which the tool axis is currently pointed using the axis direction keys or the handwheel.

Use this function if

- You want to retract the tool in the direction of the tool axis during suspension of a 5-axis machining program
- You want to machine with an inclined tool using the handwheel or the axis direction keys in Manual Operation mode

▶ To select manual tilting, press the **3-D ROT** soft key.

▶ Use the arrow keys to move the cursor to the **Manual Operation Tool Axis** menu item

▶ Press the **ACTIVE** soft key

▶ Press the **END** key
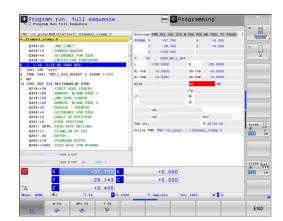
To deactivate, set the **Manual Operation Tool Axis** menu item to **Inactive** in the Tilt working plane menu.

The 🛠 icon appears in the status display when the Move in tool axis direction function is active.

Setting a preset in a tilted coordinate system

After you have positioned the rotary axes, set the preset in the same manner as for a non-tilted system. The behavior of the control during presetting depends on the setting in the optional machine parameter **chkTiltingAxes** (no. 204601):

Further information: "Introduction", Page 206

5.12 Camera-based Visual Setup Control VSC (Option 136)

Basics

Application

> ⚙ Refer to your machine manual!
> This function must be enabled and adapted by the machine tool builder.

Camera-based setup control (Option 136 Visual Setup Control) can be used to monitor the current setup situation before and during machining and compare it to a safe desired condition. After setup, simple cycles for automatic monitoring are available.

A camera system generates reference images of the current working space. With Cycles 600 **GLOBAL WORKING SPACE** or 601 **LOCAL WORKING SPACE**, the control produces an image of the working space and compares the image with previously prepared reference images. These cycles can highlight irregularities in the working space. The operator decides whether the NC program is interrupted in the event of an error or continues to run.

Using VSC offers the following advantages:

- The control can recognize elements (e.g. tools, fixtures, etc.) that are in the working space once the program has started
- If you always want to clamp a workpiece at the same position (e.g. hole at top right), the control can check the setup situation
- For documentation purposes you can generate an image of the current workspace (e.g. of a clamping situation that is rarely used)

Further information: Cycle Programming User's Manual

Requirements

As well as option number 136, a HEIDENHAIN camera system is required for VSC functions.

You must create an adequate number of reference images to allow the system to compare the situation reliably.

Overview
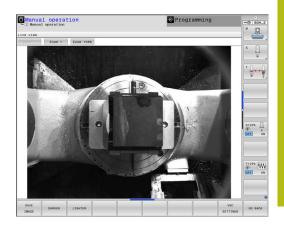
In the **Manual operation** mode, the control offers the following options:

Soft key	Function
CAMERA	Open main VSC menu
LIVE IMAGE	Show current camera view Produce live image
MONITORING DATA MANAGEMENT	Open VSC file manager The control shows the data saved for Cycle 600 and Cycle 601.
OPEN THE CAMERA COVER	Open camera cover
CLOSE THE CAMERA COVER	Close camera cover

Producing a live image

In the **Manual operation** mode, you can display and save the current camera view as a live image.

The control does not use the image captured here for automatic checking of the clamping situation. Images produced in this menu may be used for documentation and traceability. For example, you could record the current setup situation. The control saves the image produced as a .png file in the selected target directory.

Procedure

Proceed as follows to save the camera's live image:

▶ Press the **CAMERA** soft key

▶ Press the **LIVE IMAGE** soft key
> The control displays the current camera view.
> The control opens a pop-up window.
▶ Enter the desired file name
▶ Select the desired target directory
▶ Press the **OK** soft key
> The control saves the current live image.
▶ Alternative: Press the **Save** button

Options in Live Image mode

The control provides the following options:

Soft key	Function
LIGHTER	Increase camera brightness The settings made here only affect Live Image mode. They have no influence on pictures taken in automatic mode.
DARKER	Reduce camera brightness The settings made here only affect Live Image mode. They have no influence on pictures taken in automatic mode.
VSC SETTINGS	Configure the field of view of the camera Refer to your machine manual! These settings can only be made after entering a code number.
GO BACK	Go back to the previous screen

Managing monitoring data

In the **Manual operation** mode you can manage images from Cycles 600 and 601.

Proceed as follows to enter the monitoring data:

CAMERA	▶ Press the **CAMERA** soft key

MONITORING DATA MANAGEMENT	▶ Press the **MONITORING DATA MANAGEMENT** soft key > The control shows a list of the NC programs being monitored.
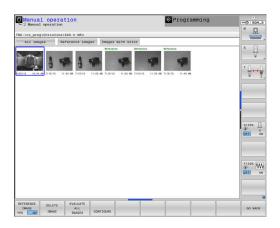
OPEN	▶ Press the **OPEN** soft key > The control shows a list of monitoring points. ▶ Edit the desired data

Selecting data

You can select the buttons with the mouse. These interfaces make it easier to search and show results in a manageable way.

- **All images**: Display all images for this monitoring file
- **Reference images**: Only display reference images
- **Images with error**: Display all images where you have highlighted an error

Features of the monitoring data management

Soft key	Function
REFERENCE IMAGE YES NO	Mark selected image as a reference image
	Please note: A reference image shows a situation in the working space that you regard as safe.
	All reference images are taken into account for the evaluation. If you add or remove an image as a reference image, this has an effect on the results of image evaluation.
DELETE IMAGE	Delete image currently selected
EVALUATE ALL IMAGES	Carry out automatic image evaluation
	The control carries out an image evaluation according to the reference images and the monitoring areas.
CONFIGURE	Change monitoring area or highlight an error
GO BACK	Go back to the previous screen
	If you change the configuration, the control carries out an image evaluation.

Configuration

You have the option of changing your settings for the monitoring area and errors at any time. When you press the **CONFIGURE** soft key, the soft-key row is switched and you can change your settings.

Soft key	Function
CONFIGURE	Change settings for the monitoring area and sensitivity If you make a change in this menu, the results of the image evaluation may change.
DRAW AREA	Draw new monitoring area If you add a new monitoring area, or change/delete monitoring areas already set, this will have an effect on the image evaluation. The same monitoring area applies to all reference images.
DRAW ERROR	Draw new error
EVALUATE IMAGE	The control checks if or how the new settings affect this image
EVALUATE ALL IMAGES	The control checks if or how the new settings affect all images
SHOW AREAS	The control shows all drawn monitoring areas
SHOW COMPARISON	The control compares the momentary image with the mean image
SAVE AND GO BACK	Save current image and return to the previous screen If you change the configuration, the control carries out an image evaluation.
GO BACK	Discard images and return to the previous screen

You can also zoom the image with the buttons and shift the magnified image section with the mouse or arrow keys.

Drawing the monitoring area or error area

Proceed as follows:

▶ Press the appropriate soft key, e.g. **DRAW AREA**

▶ Click on the image and select the area with the mouse

▷ The control indicates the clicked area with a frame.

▶ Shift the area if required by holding down the mouse button

You can fix the drawn area by double-clicking it, thereby protecting it from unintentional shifting.

Deleting drawn areas

If you have drawn several monitoring areas or error areas, you can delete these individually.

Proceed as follows:
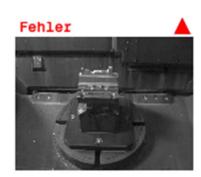
▶ Click on the area you wish to delete

> The control indicates the clicked area with a frame.

▶ Press the **Delete** button

Results of the image evaluation
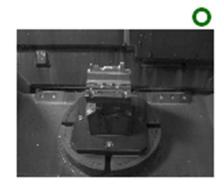
The results of the image evaluation depend on the monitoring area and the reference images. When evaluating all images, each image is evaluated according to the current configuration and the results are compared with the data last saved.

If you change the monitoring area or add/delete reference images, the images may be tagged with the following symbol:

- **Triangle:** You have changed the monitoring data, e.g. tagged an image with errors as a reference image or deleted a monitoring area. This makes the monitoring less sensitive.

  This has an effect on your reference images and on the mid-value image. As a result of your change to the configuration, the control can no longer detect errors that had been saved for this image! If you would like to continue, confirm the reduced monitoring sensitivity and the new settings will be accepted.

- **Solid circle:** You have changed the monitoring data, monitoring is more sensitive.

- **Blank circle:** No error message: All deviations saved in the image have been recognized, monitoring has not identified any conflicts.

6

Testing and running

6.1   Graphics

Application

The control graphically simulates a machining operation in the following operating modes:

- **Manual operation**
- **Program run, single block**
- **Program run, full sequence**
- **Test Run**
- **Positioning w/ Manual Data Input**

> **i** In **Positioning w/ Manual Data Input** operating mode, the workpiece blank that is currently active in the **Program Run, Full Sequence/Single Block** operating mode is shown.

The graphic depicts the workpiece as if it were being machined with tool.

If you select the **PROGRAM + MACHINE** screen layout, the control displays the defined workpiece, the collision objects, and a tool.

For active tool tables, the control also takes the entries in the columns **L**, **LCUTS**, **T-ANGLE** and **R2** into consideration.

The control will not show a graphic if

- no NC program is selected
- the wrong screen layout is selected
- the current NC program has no valid workpiece blank definition
- with blank form definition with a subprogram, the BLK FORM block was not yet run

> **i** The simulation of NC programs with 5-axis machining or tilted machining might run at reduced speed. With the MOD menu **Graphic settings**, you can decrease the **Model quality** and in that way increase the speed of simulation.

> **⚙** If you are using a TNC 640 with touch control, you can replace some keystrokes with gestures.
> **Further information:** "Operating the touchscreen", Page 533

VIEW OPTIONS

Proceed as follows to display the **VIEW OPTIONS** :

▶ Select the desired operating mode

VIEW OPTIONS

       ▶ Press the **VIEW OPTIONS** soft key

The available soft keys depend on the following settings:

- The selected screen layout.
  You can select the screen layout with the **SCREEN LAYOUT** key.
- The selected view.
  You can select the view with the **VIEW** soft key.
- The selected model quality.
  You can select the model quality in the **Graphic settings** MOD function.

The control provides the following **VIEW OPTIONS**:

Soft keys	Function
MACHINE	Show the collision objects and the workpiece
WORKPIECE	Show the workpiece
TOOL	Show the tool **Further information:** "Tool", Page 259
TOOL PATHS	Show the tool paths **Further information:** "Tool", Page 259
VIEW	Select the view **Further information:** "View", Page 260
RESET TOOL PATHS	Reset the tool paths
RESET BLK FORM	Reset the workpiece blank
BLANK FRAME OFF ON	Show the workpiece blank frame
WORKPIECE EDGES OFF ON	Highlight the workpiece edges on 3-D model
SHOW BLOCK NO. OFF ON	Show the block numbers of the tool paths
MARK END POINT OFF ON	Show the end points of the tool paths

Soft keys	Function
WORKPIECE IN COLOR OFF ON	Show the workpiece in color
REMOVE CHIPS	Remove the chips Parts of the workpiece material that are cut off by machining will be removed from the graphics.
RESET TOOL PATHS	Reset the tool paths
	Rotate and zoom the workpiece **Further information:** "Rotating, zooming and moving a graphic", Page 261
	Shift the sectional plane in the projection in three planes **Further information:** "Shifting the sectional plane", Page 263

Operating notes:

- With the machine parameter **clearPathAtBlk** (no. 124203), you specify whether the tool paths are cleared in the **Test Run** operating mode with a new BLK FORM.
- If points were output incorrectly by the postprocessor, machining marks may occur on the workpiece. To recognize these unwanted machining marks in time (prior to machining), you can test externally created NC programs for corresponding irregularities by displaying the tool paths.
- The control saves the state of the soft keys in non-volatile memory.

Tool

Displaying the tool

If the **L** and **LCUT** columns are defined in the tool table, the tool is graphically displayed.

Further information: "Entering tool data into the table", Page 137

The control displays the tool in various colors:

- Turquoise: tool length
- Red: length of cutting edge and tool is engaged
- Blue: length of cutting edge and tool is retracted

> **i** If the **ZL** and **XL** columns are defined in the turning tool table, the indexable insert is displayed and the base body is shown schematically.
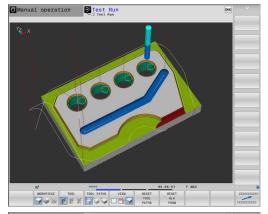
Showing the tool paths
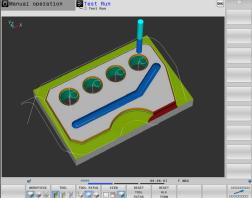
The control displays the following types of movement:

Soft keys	Function
TOOL PATHS	Movements at rapid traverse and at the programmed feed rate
TOOL PATHS	Movements at the programmed feed rate
TOOL PATHS	No movements

> **i** If you move at rapid traverse inside a workpiece, both the movement and the workpiece are shown in red at the respective location.
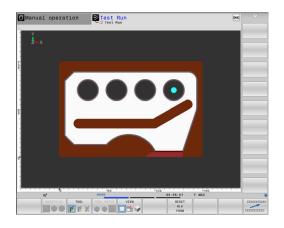
View
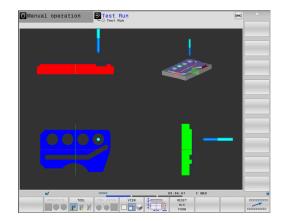
The control provides the following views:

Soft keys	Function
VIEW	Plan view
VIEW	Projection in three planes
VIEW	3-D view
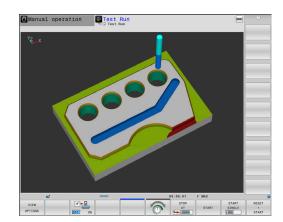
Projection in three planes

The simulation shows three sectional planes and a 3-D model, similar to a technical drawing.

3-D view

The high-resolution 3-D view enables you to display the surface of the machined workpiece in greater detail. Using a simulated light source, the control creates realistic light and shadow conditions.

Rotating, zooming and moving a graphic

Proceed as follows to rotate a graphic, for example:

- ▶ Select functions for rotating and zooming
- > The control displays the following soft keys.

Soft keys		Function
		Rotate in 5° steps about the vertical axis
		Tilt in 5° steps about the horizontal axis
		Enlarge the graphic stepwise
		Reduce the graphic stepwise
		Reset the graphic to its original size and angle
		Move the graphic upward or downward
		Move the graphic to the left or right
		Reset the graphic to its original position and angle

You can also use the mouse to change the graphic display. The following functions are available:

- ▶ In order to rotate the model shown in three dimensions, hold down the right mouse button and move the mouse. If you simultaneously press the shift key, you can only rotate the model horizontally or vertically
- ▶ To shift the model shown: Hold the center mouse button or mouse wheel down and move the mouse. If you simultaneously press the shift key, you can only shift the model horizontally or vertically
- ▶ To zoom in on a certain area: Mark a zoom area by holding the left mouse button down.
- > After you release the left mouse button, the control zooms in on the defined area.
- ▶ To rapidly magnify or reduce any area: Rotate the mouse wheel backwards or forwards
- ▶ To return to the standard display: Press the shift key and simultaneously double-click with the right mouse key. The rotation angle is maintained if you only double-click with the right mouse key

Setting the speed of the Test Run mode

> ℹ The most recently set speed stays active until a power interruption. After the control is switched on the speed is set to FMAX.

After you have started a program, the control displays the following soft keys with which you can set the simulation speed:

Soft key	Functions
1:1	Test the NC program with the speed that will be used when actually running the program (programmed feed rates will be taken into account)
	Increase the simulation speed incrementally
	Decrease the simulation speed incrementally
MAX	Test run at the maximum possible speed (default setting)

You can also set the simulation speed before you start a program:

▶ Select the function for setting the simulation speed

▶ Select the desired function by soft key, e.g. incrementally increasing the simulation speed

Repeating graphic simulation

A part program can be graphically simulated as often as desired. To do so you can reset the graphic to the workpiece blank.

Soft key	Function
RESET BLK FORM	Display the unmachined workpiece blank in the **Program run, single block** and **Program run, full sequence** operating modes
RESET THE VOLUME MODEL	Display the unmachined workpiece blank in the **Test Run** operating mode

Shifting the sectional plane

The default setting of the sectional plane is selected so that it lies in the working plane in the center of the workpiece blank and in the tool axis on the top surface of the workpiece blank.

Proceed as follows to move the sectional plane:

> ▶ Press the soft key for
>   **shifting the sectional plane**

> ⊳ The control displays the following soft keys:

Soft keys		Function
		Shift the vertical sectional plane to the right or left
		Shift the vertical sectional plane forward or backward
		Shift the horizontal sectional plane upwards or downwards

The position of the sectional planes is visible during shifting. The shift remains active, even if you activate a new workpiece blank.

Resetting sectional planes

The shifted sectional plane also remains active for a new workpiece blank. The sectional plan is automatically reset when the control is restarted.

Proceed as follows to move the sectional plane to the default position:

> ▶ Press the soft key for
>   **resetting the sectional planes** soft key

6.2 Measurement of machining time

Machining time in the Test Run operating mode

The control calculates the duration of the tool movements and displays this as machining time in the test run. The control takes feed movements and dwell times into account.

The time determined by the control is only of limited value for calculating the machining time because it does not take any machine-dependent time intervals (e.g., for tool changes) into consideration.

| **i** | The machining times determined using the graphic simulation do not correspond to the actual machining times. Reasons for this during combined milling-turning operations include the switching of operating modes. |

Proceed as follows to select the stopwatch function:

| ⊙ | ▶ Select the stopwatch function |
| STORE ⊙ | ▶ Select the desired function by soft key, e.g. saving the displayed time |
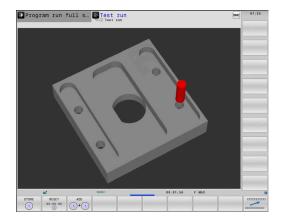
Soft key	**Stopwatch functions**
STORE ⊙	Save the displayed time
ADD ⊙+⊙	Display the sum of saved time and displayed time
RESET 00:00:00 ⊙	Clear the displayed time

Machining time in the machine operating modes

Time display from program start to program end. The timer stops whenever machining is interrupted.

6.3 Showing the workpiece blank in the working space

Application

In **Test Run** operating mode, you can graphically check the position of the workpiece blank and the preset in the working space of the machine. The graphic shows the preset that has been set in the NC program using Cycle 247. If you have not defined a preset in the NC program, then the graphic shows the active preset on the machine.

A transparent cuboid represents the workpiece blank. Its dimensions are shown in the **BLK FORM** table. The control takes over the dimensions from the workpiece blank definition of the selected NC program.
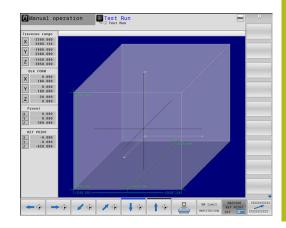
For a test run it normally does not matter where the workpiece blank is located within the working space. If you activate **BLANK IN WORK SPACE** working space monitoring, you must graphically shift the workpiece blank in a way that it lies within the working space. Use the soft keys shown in the table.

In addition, you can use the current machine status for the **Test Run** operating mode.

The current machine status includes the following:

- active machine kinematics
- active traverse ranges
- active machining modes
- active working spaces
- active preset

Soft keys		Function
← ⊕	→ ⊕	Shift the workpiece blank in positive or negative X direction
↙ ⊕	↗ ⊕	Shift the workpiece blank in positive or negative Y direction
↑ ⊕	↓ ⊕	Shift the workpiece blank in positive or negative Z direction
🔧		Use the current machine status
ACTIVE TRAVERSE RANGES		Display the current traverse range
SELECT TRAVERSE RANGE		Selecting the traverse range
		The ranges of traverse are configured by the machine manufacturer.

Soft keys	Function
SW-ENDSCH. ÜBERWACHUNG AUS [EIN]	Switch on/off the monitoring function
MACHINE REF POINT OFF [ON]	Display the machine preset
BEZUGSPKT. ZURÜCK- SETZEN	Set the principal axis values of the active preset to 0 for the simulation

> **i**   For blank in work space, the control depicts the **BLK FORM** only schematically.
> - With **BLK FORM CYLINDER**, a cuboid is depicted as the workpiece blank
> - With **BLK FORM ROTATION**, no workpiece blank is depicted

6.4 Measuring

Application

In **Test Run** operating mode, you can use the **MEASURING** soft key to display the following information:

- Approximate coordinates as XYZ values
- Optional display
    - FMAX: If the control machines at the maximum feed rate.
    - Threads: If a thread cutting cycle has been programmed. (Option 50)
    - Residual material: If a contour follow-up has been programmed.
- Tool number
- Tool name

Proceed as follows to select the measuring function:

MEASURING
OFF   ON
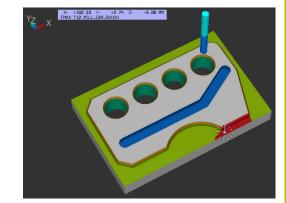
- ▶ Set the **MEASURING** soft key to **ON**

- ▶ Position the mouse pointer on the respective position
- \> The control displays the positioning ball icon and the surface orientation with a black and white circle and a perpendicular line on it.
- \> The corresponding information is displayed in a blue text field.

> ⓘ The **MEASURING** soft key is available in the following views:
> - Plan view
> - 3-D view
>
> **Further information:** "View", Page 260

6.5 Optional program-run interruption

Application
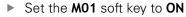
> ⚙ Refer to your machine manual!
> The behavior of this function varies depending on the respective machine.
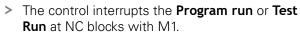
The control optionally interrupts program run at NC blocks in which an M1 has been programmed. If you use M1 in the **Program run** operating mode, the control does not switch off the spindle or the coolant.

[M01 ON OFF]	▶ Set the **M01** soft key to **OFF**
	> The control does not interrupt the **Program run** or **Test Run** at NC blocks that contain an M1.
[M01 ON OFF]	▶ Set the **M01** soft key to **ON**
	> The control interrupts the **Program run** or **Test Run** at NC blocks with M1.

6.6 Skipping NC blocks

You can have NC blocks skipped in the following operating modes:

- **Test Run**
- **Program run, full sequence**
- **Program run, single block**
- **Positioning w/ Manual Data Input**

> Operating notes:
> - This function does not work for **TOOL DEF** blocks.
> - After a power interruption the control returns to the most recently selected setting.
> - The setting of the **HIDE** soft key is only effective in the respective operating mode.

Test Run and Program Run

Application

In **Test Run** or **Program Run, Full Sequence/Single Block** operating mode, the control can skip over NC blocks that you have prefaced with a slash (**/**):

▶ Set the **HIDE** soft key to **ON**
> The control skips the NC blocks.

▶ Set the **HIDE** soft key to **OFF**
> The control runs or tests the NC blocks.

Procedure

You can optionally hide NC blocks.

Proceed as follows in order to hide NC blocks in the **Programming** mode:

▶ Select the desired NC block

▶ Press the **INSERT** soft key
> The control inserts a slash (/).

Proceed as follows in order to show NC blocks again in the **Programming** mode:

▶ Select the hidden NC block

▶ Press the **REMOVE** soft key
> The control removes the slash (/).

Positioning w/ Manual Data Input

Application

🛈   In order to skip NC blocks in the **Positioning w/ Manual Data Input** operating mode, you need an alphabetic keyboard.

In **Positioning w/ Manual Data Input** operating mode, you can have the control skip NC blocks that have been marked correspondingly:

▶ Set the **HIDE** soft key to **ON**
> The control skips the NC blocks.

▶ Set the **HIDE** soft key to **OFF**
> The control runs the NC blocks.

Procedure

Proceed as follows to hide NC blocks in the **Positioning w/ Manual Data Input** operating mode:

▶ Select the desired NC block

▶ Press the **/** key on the alphabetic keyboard
> The control inserts a slash (**/**).

Proceed as follows to show NC blocks again in the **Positioning w/ Manual Data Input** operating mode:

▶ Select the hidden NC block

▶ Press the **Backspace** key
> The control removes the slash (**/**).

6.7 Test run

Application

In the **Test Run** operating mode, you can simulate NC programs and program sections in order to reduce programming errors during program run. The control checks the programs for the following:

- Geometrical incompatibilities
- Missing data
- Impossible jumps
- Violation of the machine's working space
- Using disabled tools

The following functions are also available:

- Blockwise test run
- Interruption of test at any NC block
- Skip NC blocks
- Functions for graphic simulation
- Measure machining time
- Additional status display

Keep the following in mind when performing a test run

With cuboid workpiece blanks, the control starts a test run after a tool call at the following position:

- In the working plane in the center of the defined **BLK FORM**
- In the tool axis, 1 mm above the **MAX** point defined in the **BLK FORM**

With rotationally symmetrical workpiece blanks, the control starts a test run after a tool call at the following position:

- In the machining plane at the position X=0, Y=0
- In the tool axis 1 mm above the defined workpiece blank

The **FN 27: TABWRITE** and **FUNCTION FILE** functions are only considered in the **Program run, single block** and **Program run, full sequence** operating modes.

NOTICE

Danger of collision!

In the **Test Run** operating mode, the control does not take all axis movements of the machine into consideration (e.g., PLC positioning movements as well as movements from tool-change macros and M functions). This can cause a test performed without errors to later deviate from the machining operation. Danger of collision during machining!

- ▶ Test the NC program at the later machining position (**BLANK IN WORK SPACE**)
- ▶ Program a safe intermediate position after the tool change and before prepositioning
- ▶ Carefully test the NC program in the **Program run, single block** operating mode
- ▶ If possible, use the **Dynamic Collision Monitoring (DCM)** function

---

⚙   Refer to your machine manual!

Your machine tool builder can also define a tool-change macro for the **Test Run** operating mode. This macro will simulate the exact behavior of the machine.

In doing so, the machine tool builder often changes the simulated tool change position.

Test run execution

	For the test run, you must activate a tool table (status S). Select a tool table via the file manager in the **Test Run** operating mode.
i	For turning tools you can select a turning tool table with the extension .trn, which is compatible with the selected tool table. To do this, the turning tools must match in both selected tables.

You can select any preset table (status S) for the test run.

Once you press the **RESET + START** soft key in the **Test Run** operating mode, the control automatically uses the active preset from the machine operating modes for simulation. This preset remains selected when starting the test run until you define another preset in the NC program. The control reads all other defined presets from the preset table selected for the test run.

With the **BLANK IN WORK SPACE** function, you can activate workspace monitoring for the test run.

Further information: "Showing the workpiece blank in the working space ", Page 265

▶ Operating mode: Press the **Test Run** key

▶ Call the file manager with the **PGM MGT** key and select the file you wish to test

The control then displays the following soft keys:

Soft key	Functions
RESET + START	Reset the blank form, reset the previous tool data and test the entire NC program
START	Test the entire NC program
START SINGLE	Test each NC block individually
STOP AT	Executes the **Test Run** until NC block N
STOP	Stop test run (this soft key only appears if you have started the test run)

You can interrupt and continue the test run at any time, even within fixed cycles. In order to continue the test, the following actions must not be performed:

- Selecting another NC block with the arrow keys or the **GOTO** key
- Making changes to the NC program
- Selecting another NC program

Executing a Test Run up to a certain NC block

With the **STOP AT** function the control executes a **Test Run** up to the NC block with block number **N**.

Proceed as follows to stop the **Test Run** at any NC block:

[STOP AT icon]

- ▶ Press the **STOP AT** soft key

- ▶ **Stop at: N =** Enter the block number at which the simulation should stop
- ▶ Enter the **Program** name of the NC program containing the NC block with the selected block number
- > The control shows the name of the selected NC program.
- ▶ If the simulation is to be stopped in an NC program that has been called using **PGM CALL**, then enter this name
- ▶ **Repetitions =** If **N** is located in a program-section repeat, enter the number of repeats that you want to run.
  Default 1: The control stops before **N** is simulated
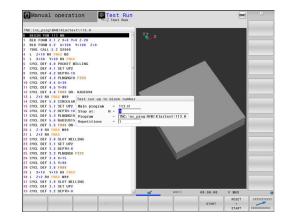
Possibilities in stopped condition

If you interrupt the **Test Run** with the **STOP AT** function, you have the following possibilities in this stopped condition:

- ▪ Enable or disable **NC block skip**
- ▪ **Optional program stop** enable or disable
- ▪ Modify graphics resolution and model
- ▪ Modify the NC program in the **Programming** operating mode

If you modify the NC program in the **Programming** operating mode, the simulation behaves as follows:

- ▪ Modification before the interruption point: The simulation restarts at the beginning
- ▪ Modification after the interruption point: Positioning at the interruption point is possible with **GOTO**

GOTO function

Using the GOTO key

Jumping with the GOTO key

Use the **GOTO** key to jump to a specific location in the NC program, regardless of the active operating mode.

Proceed as follows:

▶ Press the **GOTO** key

> The control opens a pop-up window.

▶ Enter a number

▶ Select the jump statement by soft key, e.g. move down the number of lines entered

The control provides the following options:

Soft key	Function
N LINES	Move up the number of lines entered
N LINES	Move down the number of lines entered
GOTO LINE NUMBER	Jump to the block number entered

> Use the **GOTO** function only during programming and testing of NC programs. Use the block scan function during program run.
>
> **Further information:** "Any entry into the NC program: Block scan", Page 290

Quick selection with the GOTO key

With the **GOTO** key, you can open the Smart Select window that makes it easy for you to select special functions or cycles.

Proceed as follows to select special functions:

▶ Press the **SPEC FCT** key

▶ Press the **GOTO** key

> The control displays a pop-up window showing a structural view of the special functions

▶ Select the desired function

Further information: Cycle Programming User's Manual

Opening the selection window with the GOTO key

When the control provides a selection menu, you can use the **GOTO** key to open the selection window. This allows you to view the available entries.

Display of NC programs

Syntax highlighting

The control displays syntax elements with various colors according to their meaning. Color-highlighting makes the NC programs easier to read and clearer.

Color highlighting of syntax elements

Use	Color
Standard color	Black
Display of comments	Green
Display of numerical values	Blue
Display of the block number	Violet
Display of FMAX	Orange
Display of the feed rate	Brown

Scrollbar

Screen content can be shifted with the mouse using the scroll bar at the right edge of the program window. In addition, the size and position of the scrollbar indicates program length and cursor position.

6.8 Program run

Application

In the **Program run, full sequence** operating mode, the control executes an NC program continuously to its end or up to a program stop.

In the **Program run, single block** operating mode, the control executes each NC block individually after pressing the **NC Start** key. With point pattern cycles and **CYCL CALL PAT** the control stops after each point.

You can use the following control functions in the **Program run, single block** and **Program run, full sequence** operating modes:

- Interrupt program run
- Start the program run from a certain NC block
- Skip NC blocks
- Edit the tool table TOOL.T
- Edit the active datum table or compensation table
- Checking and changing Q parameters
- Superimpose handwheel positioning
- Functions for graphic simulation
- Additional status display

Running an NC program

Preparation

1 Clamp the workpiece to the machine table.
2 Set the preset
3 Select the necessary tables and pallet files (status M)
4 Select the NC program (status M)

	Operating notes:
i	- You can change the feed rate and spindle speed using the potentiometers.
	- You can reduce the feed rate using the **FMAX** soft key. This reduction affects all rapid traverse and feed movements, even after the control has been restarted.
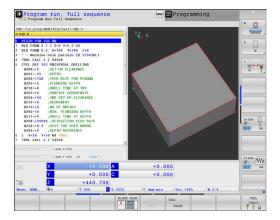
Program Run, Full Sequence

► Start the NC program with the **NC start** key

Program Run, Single Block

► Start each NC block of the NC program individually with the **NC start** key

Structuring NC programs

Definition and applications

The control enables you to comment NC programs in structuring blocks. Structuring blocks are texts with up to 252 characters and are used as comments or headlines for the subsequent program lines.

With the aid of appropriate structuring blocks, you can organize long and complex NC programs in a clear and comprehensible manner.

This function is particularly convenient if you want to change the NC program later. Structuring blocks can be inserted into the NC program at any point.

Structure blocks can also be displayed in a separate window, and be edited or added to, as desired. Use the appropriate screen layout for this.

The control manages the inserted structure items in a separate file (extension: .SEC.DEP). This speeds navigation in the program structure window.

The **PROGRAM + SECTS** screen layout can be selected in the following operating modes:

- **Program run, single block**
- **Program run, full sequence**
- **Programming**

Displaying the program structure window / Changing the active window
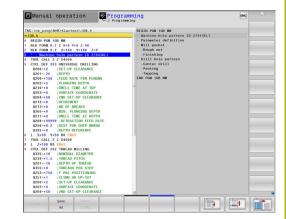
PROGRAM + SECTS
- ▶ Display structure window: For this screen layout press the **PROGRAM + SECTS** soft key

- ▶ Change the active window: Press the **CHANGE WINDOW** soft key

Selecting blocks in the program structure window

If you are scrolling through the program structure window block by block, the control at the same time automatically moves the corresponding NC blocks in the program window. This way you can quickly skip large program sections.

Checking and changing Q parameters

Procedure

You can check Q parameters in all operating modes, and also edit them.
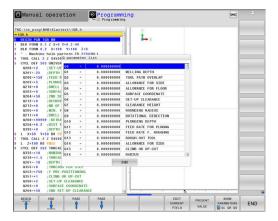
▶ If you are in a program run, interrupt it if required (e.g. by pressing the **NC STOP** key and the **INTERNAL STOP** soft key) or stop the test run


```
Q
INFO
```
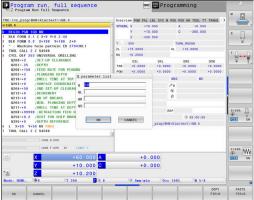
▶ To call the Q parameter functions, press the **Q INFO** soft key or the **Q** key

> The control lists all of the parameters and their corresponding current values.

▶ Use the arrow keys or the **GOTO** key to select the desired parameter.

▶ To change the value, press the **EDIT CURRENT FIELD** soft key, enter the new value, and confirm with the **ENT** key

▶ To leave the value unchanged, press the **PRESENT VALUE** soft key or close the dialog with the **END** key

ℹ All of the parameters with displayed comments are used by the control within cycles or as transfer parameters.

If you want to check or edit local, global or string parameters, press the **SHOW PARAMETERS Q QL QR QS** soft key. The control then displays the specific parameter type. The functions previously described also apply.

You can have Q parameters also displayed in the additional status display in all operating modes (except **Programming** mode).

▶ If you are in a program run, interrupt it if required (e.g. by pressing the **NC STOP** key and the **INTERNAL STOP** soft key) or stop the test run

⬡	▶ Display the soft key row for screen layout

PROGRAM + STATUS	▶ Select the layout option for the additional status display
	> In the right half of the screen, the control shows the **Overview** status form.

STATUS OF Q PARAM.	▶ Press the **STATUS OF Q PARAM.** soft key

Q PARAMETER LIST	▶ Press the **Q PARAMETER LIST** soft key
	> The control opens a pop-up window.
	▶ For each parameter type (Q, QL, QR, QS), define the parameter numbers you wish to check. Separate single Q parameters with a comma, and connect sequential Q parameters with a hyphen, e.g. 1,3,200-208. The input range per parameter type is 132 characters

> **ℹ** The display in the **QPARA** tab always contains eight decimal places. The result of Q1 = COS 89.999 is shown by the control as 0.00001745, for example. Very large or very small values are displayed by the control in exponential notation. The result of Q1 = COS 89.999 * 0.001 is shown by the control as +1.74532925e-08, whereby e-08 corresponds to the factor of 10^{-8}.

Interrupting, stopping or aborting machining

There are several ways to stop a program run:

- Interrupt the program run with e.g. the miscellaneous function **M0**
- Interrupt the program run e.g. with the **NC stop** soft key
- Stop the program run, e.g. with the **NC stop** key in connection with the **INTERNAL STOP** soft key
- Terminate the program run e.g. with the miscellaneous functions **M2** or **M30**

The control shows the current status of the program run in the status display.

Further information: "General status display", Page 68

In contrast to a stopped run, an interrupted, aborted (terminated) program run enables certain actions by the user, including the following:

- Select operating mode
- Check Q parameters and change these if necessary using the **Q INFO** function
- Change setting for the optional programmed interruption with **M1**
- Change setting for the programmed skipping of NC blocks with **/**

> **i**   During major errors, the control automatically aborts the program run (e.g., during a cycle call with stationary spindle).

Program-controlled interruptions

You can set interruptions directly in the NC program. The control interrupts the program run in the NC Block containing one of the following inputs:

- Programmed stop **STOP** (with and without miscellaneous function)
- Programmed stop **M0**
- Conditional stop **M1**

NOTICE

Danger of collision!

Certain manual interactions cause the control to lose program information affecting the mode and thereby to lose the so-called contextual reference. After the loss of the contextual reference, unexpected and undesired movements can occur. There is a danger of collision during subsequent machining operations!

- ▶ Do not perform the following interactions:
  - Cursor movement to another NC block
  - The jump command **GOTO** to another NC block
  - Editing an NC block
  - Modifying Q parameter values with the **Q INFO** soft key
  - Switching the operating modes
- ▶ Restore the contextual reference via repetition of the required NC blocks

---

⚙ Refer to your machine manual!

The miscellaneous function **M6** may also lead to a suspension of the program run. The machine manufacturer sets the functional scope of the miscellaneous functions.

Manual program interruption

While an NC program is being executed in the **Program run, full sequence** operating mode, select the **Program run, single block** operating mode. The control interrupts the machining process at the end of the current machining step.

Aborting program run

▶ Press the **NC Stop** key
> The control does not finish the current NC block.
> The control shows the icon for stopped status in the status display.
> Actions such as a change of operating mode are not possible.
> The program can be resumed with the **NC start** key.

▶ Press the **INTERNAL STOP** soft key

> The control briefly shows the icon for program cancellation in the status display.
> The control shows the icon for the exited inactive status in the status display.
> Actions such as a change of operating mode are available again.

Compensation during the program run

During the program run, you can access the programmed compensation tables and the active datum table. It is even possible to make changes to these tables.

The changes, however, only become effective after you have activated the corresponding compensation table again.

To access a table, proceed as follows:

▶ Press the **SELECT COMPENS. TABLES** soft key

▶ Press the soft key for the desired table, e.g. **DATUM TABLE**
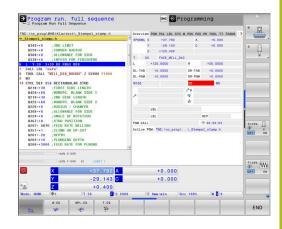
Further information: User's Manuals for Conversational Programming or ISO Programming

Moving the machine axes during an interruption

During program-run interruption, you can move the axes manually. If, at the time of the interruption, the **Tilt the working plane** function is active, then the **3-D ROT** soft key becomes available.

In the **3-D ROT** menu, you can choose between the following functions:

Soft key	Status display icon	Function
	No icon	You can move the axes in the machine coordinate system (M-CS).
		Further information: "Machine coordinate system M-CS", Page 117
W-CS		You can move the axes in the workpiece coordinate system (W-CS).
		Further information: "Workpiece coordinate system W-CS", Page 120
WPL-CS		You can move the axes in the working-plane coordinate system (WPL-CS).
		Further information: "Working plane coordinate system WPL-CS", Page 122
T-CS		You can move the axes in the tool coordinate system (T-CS).
		The control locks the other axes.
		Further information: "Tool coordinate system T-CS", Page 125

> Refer to your machine manual!
>
> Your machine tool builder enables the function for traversing in the tool-axis direction.

NOTICE

Danger of collision!

During a program interruption, you can move the axes manually (e.g., in order to retract from a hole when the working plane is tilted). There is a risk of collision if the **3-D ROT** setting is incorrect!

▶ It is better to use the **T-CS** function
▶ Use a low feed rate

Modifying the preset during an interruption

If you modify the active preset during an interruption, resuming the program run is only possible with **GOTO** or mid-program startup at the interruption point.

Example: Retracting the spindle after tool breakage

▶ Interrupt machining

▶ Enable the axis direction keys: Press the **MANUAL TRAVERSE** soft key

▶ Move the machine axes with the axis direction keys

⚙   Refer to your machine manual!

On some machines you may have to press the **NC start** key after the **MANUAL TRAVERSE** soft key to enable the axis direction keys.

Resuming program run after an interruption

The control saves the following data during a program interruption:

■ The last tool that was called

■ Current coordinate transformations (e.g., datum shift, rotation, mirroring)

■ The coordinates of the circle center that was last defined

The control uses the stored data for returning the tool to the contour after manual machine axis positioning during an interruption (**RESTORE POSITION** soft key).

ℹ   Operating notes:

■ The saved data remains active until it is reset (e.g., by selecting a program).

■ If you interrupt an NC program using the **INTERNAL STOP** soft key, you have to start machining at the start of the program or by using the **BLOCK SCAN** function.

■ For program interruptions within program-section repeats or subprograms, you have to use the **BLOCK SCAN** function for re-entering at the point of interruption.

■ With machining cycles, mid-program startup is always executed at the start of the cycle. If you interrupt a program run during a machining cycle, the control repeats machining steps already carried out after a block scan.

Resuming the program run with the NC Start key

You can resume program run by pressing the **NC start** key if the NC program was stopped in one of the following ways:

■ Press the **NC Stop** key

■ Programmed interruption

Resuming program run after an error

With an erasable error message:

- ▶ Remove the cause of the error
- ▶ Clear the error message from the screen: Press the **CE** key
- ▶ Restart the program, or resume program run where it was interrupted

Retraction after a power interruption

⚙ Refer to your machine manual!
Your machine tool builder configures and enables the **Retraction** mode of operation.

With the **Retraction** mode of operation you can disengage the tool from the workpiece after an interruption in power.

If you activated a feed rate limit before a power failure, this is still active. You can deactivate the feed-rate limit using the **CANCEL THE FEED RATE LIMITATION** soft key.

The **Retraction** mode of operation is selectable in the following conditions:

- ■ Power interrupted
- ■ No control voltage for the relay
- ■ Traversing the reference points
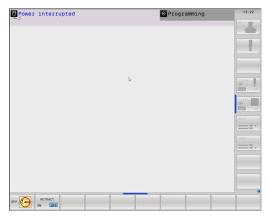
The **Retraction** operating mode offers the following modes of traverse:
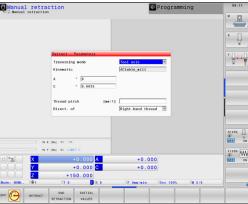
Mode	Function
Machine axes	Movement of all axes in the machine coordinate system
Tilted system	Movement of all axes in the active coordinate system
	Effective parameters: Position of the tilting axes
Tool axis	Movements of the tool axis in the active coordinate system
Thread	Movements of the tool axis in the active coordinate system with compensating movement of the spindle
	Effective parameters: Thread pitch and direction of rotation
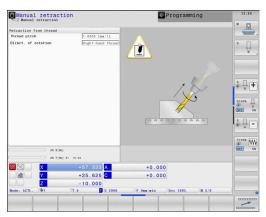
ℹ If the **Tilt the working plane** function (Option 8) is enabled on your control, then the **Tilted system** traverse mode is also available.

The control selects the mode of traverse and the associated parameters automatically. If the traverse mode oder the parameters have not been correctly preselected, you are unable to reset them manually.

NOTICE

Caution: Danger to the tool and workpiece!

A power failure during the machining operation can cause uncontrolled "coasting" or braking of the axes. In addition, if the tool was in effect prior to the power failure, then the axes cannot be referenced after the control has been restarted. For non-referenced axes, the control takes over the last saved axis values as the current position, which can deviate from the actual position. Thus, subsequent traverse movements do not correspond to the movements prior to the power failure. If the tool is still in effect during the traverse movements, then the tool and the workpiece can sustain damage through tension!

► Use a low feed rate

► Please keep in mind that the traverse range monitoring is not available for non-referenced axes

---

Example

The power failed while a thread cutting cycle in the tilted working plane was being performed. You have to retract the tap:

► Switch on the power supply for control and machine

> The control starts the operating system. This process may take several minutes.

> The control will then display the **Power interrupted** message in the screen header.

► Activate the **Retraction** mode: Press the **RETRACT** soft key

> The control displays the message **Retraction selected**

► Confirm the power interruption: Press the **CE** key

> The control compiles the PLC program.

► Switch on the machine control voltage

> The control checks the functioning of the EMERGENCY STOP circuit. If there is at least one non-referenced axis, you will have to compare the displayed position values with the actual axis values and confirm that they are correct. if required, follow the dialog.

► Check the preselected traverse mode: If required, select **THREAD**

► Check the preselected thread pitch: If required, enter the thread pitch

► Check the preselected direction of rotation: if needed, select the turning direction of the thread
Right-handed thread: the main spindle turns clockwise when moving into the workpiece, counter-clockwise when retracting from it; left-handed thread: main spindle turns counter-clockwise when moving into the workpiece and clockwise when retracting from it

RETRACT

▶ Activate retraction: Press the **RETRACT** soft key

▶ Retraction: Retract the tool with the axis direction keys or the electronic handwheel
Axis key Z+: Retraction from the workpiece
Axis key Z-: Moving into the workpiece

▶ Exit retraction: Return to the original soft-key level

END RETRACTION

▶ End the **Retraction** mode: Press the **END RETRACTION** soft key

> The control checks whether the **Retraction** mode can be ended. If necessary, follow the dialog.

▶ Answer confirmation request: If the tool was not correctly retracted, press the **NO** soft key. If the tool was correctly retracted, press the **YES** soft key.

> The control hides the **Retraction selected** dialog.

▶ Initialize the machine: if required, cross the datums

▶ Establish the desired machine condition: If required, reset the tilted working plane

Any entry into the NC program: Block scan

⚙ Refer to your machine manual!
The **BLOCK SCAN** function must be enabled and configured by the machine tool manufacturer.

With the **BLOCK SCAN** function you can start an NC program at any desired NC block. The control takes workpiece machining up to this NC block into account in its calculations.

If the NC program was interrupted under the following conditions, the control saves the interruption point:

- **INTERNAL STOP** soft key
- Emergency stop
- Power failure

If, while restarting, the control finds a saved point of interruption, then it outputs a message. You can then execute mid-program startup directly at the point of interruption.

You can run the mid-program startup in the following ways:

- Mid-program startup in the main program, with repetitions if necessary
- Multi-level mid-program startup in subprograms and touch probe cycles
- Mid-program startup in a point table
- Block scan in pallet programs

At the start of the block scan, the control resets all data, as with a selection of the NC program. During the block scan, you can switch between **Program Run Full Sequence** and **Program Run Single Block**.

NOTICE

Danger of collision!
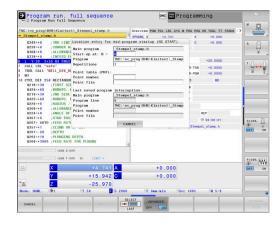
The **BLOCK SCAN** function skips over the programmed touch probe cycles. As a result, the result parameters contain no values or, possibly, incorrect values. If the subsequent machining operation uses these result parameters, then there is a risk of collision!

▶ Use the **BLOCK SCAN** function at multiple levels
Further information: "Procedure for multi-level mid-program startup", Page 292

ℹ The **BLOCK SCAN** function must not be used in conjunction with the following functions:

- Active stretch filter
- Touch probe cycles 0, 1, 3, and 4 during the search phase of mid-program startup

Procedure for simple mid-program startup

> ⓘ   The control only displays the dialogs required by the process in the pop-up window.

▶ Press the **BLOCK SCAN** soft key

> The control shows a pop-up window with the active main program.

▶ **Start-up at: N =**: Enter the number of the NC block where you wish to enter the NC program

▶ **Program**: Check the name and path of the NC program containing the NC block, or enter with the **SELECT** soft key

▶ **Repetitions**: Enter the number of repetitions which should be taken into account in the block scan if the NC block is located within a program section repeat. **Default 1 means first machining**

▶ Press the **ADVANCED** soft key if required

▶ If required, press the **INSERT LAST NC BLOCK** soft key in order to select the last saved interruption

▶ Press the **NC Start** key

> The control starts the block scan, calculates until the entered NC block and shows the next dialog.

If you changed the machine status:

▶ Press the **NC Start** key

> The control restores the machine status, e.g. TOOL CALL, M functions and shows the next dialog.

If you changed the axis positions:

▶ Press the **NC Start** key

> The control approaches the specified positions in the specified sequence and shows the next dialog.
> Approach axes in individually selected sequence:
> **Further information:** "Returning to the contour", Page 297

▶ Press the **NC Start** key

> The control resumes execution of the NC program.

Example of simple mid-program startup

ⓘ   Even after an internal stop, the control shows the number of repetitions on the Overview tab.

After an internal stop, you would like to start in NC block 12 in the third machining operation of LBL 1.

In the pop-up window enter the following data:

- **Start-up at: N =**12
- **Repetitions** 3

Procedure for multi-level mid-program startup

If you, for example, start in a subprogram that is called several times by the main program, then use the multi-level block scan. For this purpose, jump in the main program to the desired subprogram call. With the **CONTINUE BLOCK SCAN** function, you can jump further from this position.

ⓘ   Operating notes:
- The control only displays the dialogs required by the process in the pop-up window.
- You can also continue the **BLOCK SCAN** without restoring the machine status and the axis position of the first startup point. For this, press the **CONTINUE BLOCK SCAN** soft key before confirming the restoration with the **NC-Start** key.

Mid-program startup to the first start-up point:

▶ Press the **BLOCK SCAN** soft key

▶ Enter the first NC block where you wish to start

▶ Press the **ADVANCED** soft key if required

▶ If required, press the **INSERT LAST NC BLOCK** soft key in order to select the last saved interruption

▶ Press the **NC Start** key

> The control starts the block scan and calculates until the entered NC block.

If the control should restore the machine status of the entered NC block:

▶ Press the **NC Start** key

> The control restores the machine status, e.g. TOOL CALL, M functions.

If the control should restore the axis positions:

▶ Press the **NC Start** key

> The control moves in the specified sequence to the specified positions.

If the control should run the NC block:

▶ Select the **Program Run Single Block** operating mode if required

▶ Press the **NC Start** key

> The control runs the NC block.

Mid-program startup to the next start-up point:

▶ Press the **CONTINUE BLOCK SCAN** soft key

▶ Enter the NC block where you wish to start

If you changed the machine status:

▶ Press the **NC Start** key

If you changed the axis positions:

▶ Press the **NC Start** key

If the control should run the NC block:

▶ Press the **NC Start** key

▶ Repeat these steps if required to jump to the next start-up point

▶ Press the **NC Start** key

> The control resumes execution of the NC program.

Example of multi-level mid-program startup

You run a main program with several subprogram calls in the NC program Sub.h. You work with a touch probe cycle in the main program. You use the result of the touch probe cycle later for positioning.

After an internal stop you wish to start up in NC block 8 in the second call of the subprogram. This subprogram call is in NC block 53 of the main program. The touch probe cycle is in NC block 28 of the main program, i.e. before the desired start-up point.

BLOCK SCAN	▶ Press the **BLOCK SCAN** soft key
	▶ In the pop-up window enter the following data:
	■ **Start-up at: N =**28
	■ **Repetitions** 1
	▶ Select the **Program Run Single Block** operating mode if required
	▶ Press the **NC start** key until the control runs the touch probe cycle
	> The control saves the result.
CONTINUE BLOCK SCAN	▶ Press the **CONTINUE BLOCK SCAN** soft key
	▶ In the pop-up window enter the following data:
	■ **Start-up at: N =**53
	■ **Repetitions** 1
	▶ Press the **NC start** key until the control runs the NC block
	> The control jumps into the subprogram Sub.h.
CONTINUE BLOCK SCAN	▶ Press the **CONTINUE BLOCK SCAN** soft key
	▶ In the pop-up window enter the following data:
	■ **Start-up at: N =**8
	■ **Repetitions** 1
	▶ Press the **NC start** key until the control runs the NC block
	> The control continues to run the subprogram and then returns to the main program.

Block scan in a point table

If you start in a point table called by the main program, use the **ADVANCED** soft key.

▶ Press the **BLOCK SCAN** soft key

> The control shows a pop-up window.

▶ Press the **ADVANCED** soft key

> The control expands the pop-up window.

▶ **Point number**: Enter the line number of the point table you start with

▶ **Point file**: Enter the name and path of the point table

▶ If required, press the **SELECT LAST BLOCK** soft key in order to select the last saved interruption

▶ Press the **NC Start** key

If you would like to use the block scan function to start in a point pattern, then proceed just as you would for starting in the point table. Enter the desired point number in the **Point number** input field. The first point in the point pattern has the point number **0**.

Block scan in pallet programs

With the pallet management , you can also use the **BLOCK SCAN** function in conjunction with pallet tables.

If you interrupt the processing of pallet tables, the control always suggests the previously selected NC block of the interrupted NC program for the **BLOCK SCAN** function.

---

i   For **BLOCK SCAN** in pallet tables, you also define the **Pallet line** input field. The input refers to the line in the **NR** pallet table. This input is always required as an NC program may occur several times in a pallet table.

The **BLOCK SCAN** always takes place in a workpiece-oriented manner, even if you have selected the machining method **TO** and **CTO**. After the **BLOCK SCAN**, the control continues working again in accordance with the selected machining method.

▶ Press the **BLOCK SCAN** soft key

> The control shows a pop-up window.

▶ **Pallet line**: Enter the line number of the pallet table

▶ Enter the **Repetitions** if the NC block is located within a program section repeat

▶ Press the **ADVANCED** soft key if required

> The control expands the pop-up window.

▶ Press the **SELECT LAST BLOCK** soft key to select the last saved interruption

▶ Press the **NC Start** key

Returning to the contour

With the **RESTORE POSITION** function, the control moves the tool
to the workpiece contour in the following situations:
- Return to the contour after the machine axes were moved
  during a program interruption that was not performed with the
  INTERNAL STOP function.
- Return to the contour with a block scan with
  RESTORE POS AT N, for example after an interruption with
  INTERNAL STOP
- Depending on the machine, if the position of an axis has
  changed after the control loop has been opened during a
  program interruption

Procedure

Proceed as follows to approach the contour:

RESTORE POSITION	▶ Press the **RESTORE POSITION** soft key
	▶ Restore the machine status, if required

Approach the axes in the sequence shown by the control:

⟦⟧	▶ Press the **NC Start** key
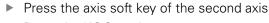
Approach the axes according to individually selected sequence:

SELECT AXIS	▶ Press the **SELECT AXIS** soft key
	▶ Press the axis soft key of the first axis
⟦⟧	▶ Press the **NC Start** key
	▶ Press the axis soft key of the second axis
⟦⟧	▶ Press the **NC Start** key
	▶ Repeat the process for all axes

> ⓘ If the tool is located in the tool axis below the starting
> point, then the control offers the tool axis as the first
> traverse direction.

6.9 Running CAM programs

If you create NC programs externally using a CAM system, you should pay attention to the recommendations detailed below. This will enable you to optimally use the powerful motion control functionality provided by the control and usually create better workpiece surfaces with shorter machining times. Despite high machining speeds, the control still achieves a very high contour accuracy. The basis for this is the HEROS 5 real-time operating system in conjunction with the **ADP** (Advanced Dynamic Prediction) function of the TNC 640. This enables the control to also efficiently process NC programs with high point densities.

From 3-D model to NC program

Here is a simplified description of the process for creating an NC program from a CAD model:

▶ **CAD: Model creation**
Construction departments prepare a 3-D model of the workpiece to be machined. Ideally the 3-D model is designed for the center of tolerance.

▶ **CAM: Path generation, tool compensation**
The CAM programmer specifies the machining strategies for the areas of the workpiece to be machined. The CAM system uses the surfaces of the CAD model to calculate the paths of the tool movements. These tool paths consist of individual points calculated by the CAM system so that each surface to be machined is approximated as nearly as possible while considering chord errors and tolerances. This way, a machine-neutral NC program is created, known as a CLDATA file (cutter location data). A postprocessor generates a machine- and control-specific NC program, which can be processed by the CNC control. The postprocessor is adapted according to the machine tool and the control. The postprocessor is the link between the CAM system and the CNC control.

▶ **Control: Motion control, tolerance monitoring, velocity profile**
The control uses the points defined in the NC program to calculate the movements of each machine axis as well as the required velocity profiles. Powerful filter functions then process and smooth the contour so that the control does not exceed the maximum permissible path deviation.

▶ **Mechatronics: Feed control, drive technology, machine tool**
The motions and velocity profiles calculated by the control are realized as actual tool movements by the machine's drive system.

Consider with post processor configuration

Take the following points into account with post processor configuration:

- Always set the data output for axis positions to at least four decimal places. This way you improve the quality of the NC data and avoid rounding errors, which can result in defects visible to the naked eye on the workpiece surface. Output of five decimal places may achieve improved surface quality for optical components as well as components with very large radii (i.e. small curvatures), for example molds for the automotive industry

- Always set the data output for the machining of surface normal vectors (LN blocks, only Klartext conversational programming) to exactly seven decimal places

- Avoid using successive incremental NC blocks because this may lead to the tolerances of the individual NC blocks being added together in the output

- Set the tolerance in Cycle 32 so that in standard behavior it is at least twice as large as the chord error defined in the CAM system Also note the information describing the functioning of Cycle 32.

- If the chord error selected in the CAM program is too large, then, depending on the respective curvature of a contour, large distances between NC blocks can result, each with large changes of direction. During machining this leads to drops in the feed rate at the block transitions. Recurring and equal accelerations (i.e. force excitation), caused by feed-rate drops in the heterogeneous NC program, can lead to undesirable excitation of vibrations in the machine structure.

- You can also use arc blocks instead of linear blocks to connect the path points calculated by the CAM system. The control internally calculates circles more accurately than can be defined via the input format

- Do not output any intermediate points on exactly straight lines. Intermediate points that are not exactly on a straight line can result in defects visible to the naked eye on the workpiece surface

- There should be exactly one NC data point at curvature transitions (corners)

- Avoid sequences of many short block paths. Short paths between blocks are generated in the CAM system when there are large curvature transitions with very small chord errors in effect. Exactly straight lines do not require such short block paths, which are often forced by the continuous output of points from the CAM system

- Avoid a perfectly even distribution of points over surfaces with a uniform curvature, since this could result in patterns on the workpiece surface

- For 5-axis simultaneous programs: avoid the duplicated output of positions if they only differ in the tool's angle of inclination

- Avoid the output of the feed rate in every NC block. This would negatively influence the control's velocity profile

Useful configurations for the machine tool operator:

- In order to improve the structure of large NC programs, use the control's structuring function

- Use the control's commenting function in order to document NC programs

- When machining holes and simple pocket geometries, use the comprehensive cycles available in the control
  Further information: Cycle Programming User's Manual
- For fits, output the contours with **RL/RR** tool radius compensation. This makes it easy for the machine operator to make necessary compensations

- Separate feed rates for pre-positioning, machining, and downfeeds, and define them via Q parameters at the beginning of the program

Example: Variable feed rate definitions

1 Q50 = 7500	FEED RATE FOR POSITIONING
2 Q51 = 750	FEED RATE FOR PLUNGING
3 Q52 = 1350	FEED RATE FOR MILLING
...	
25 L Z+250 R0 FMAX	
26 L X+235 Y-25 FQ50	
27 L Z+35	
28 L Z+33.2571 FQ51	
29 L X+321.7562 Y-24.9573 Z+33.3978 FQ52	
30 L X+320.8251 Y-24.4338 Z+33.8311	
...	
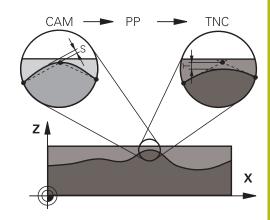
Please note the following for CAM programming

Adapting chord errors

> **i** Programming notes:
> - For finishing operations, do not set the chord error in the CAM system to a value greater than 5 μm. In Cycle 32, use an appropriate tolerance factor **T** of 1.3 to 3.
> - For roughing operations, the total of the chord error and the tolerance **T** must be less than the defined machining oversize. In this way you can avoid contour damage.
> - The specific values depend upon the dynamics of your machine.

Adapt the chord error in the CAM program, depending on the machining:

- **Roughing with preference for speed:**
  Use higher values for the chord error and the matching tolerance value in Cycle 32. Both values depend on the oversize required on the contour. If a special cycle is available on your machine, use the roughing mode. In roughing mode the machine generally moves with high jerk values and high accelerations
  - Normal tolerance in Cycle 32: Between 0.05 mm and 0.3 mm
  - Normal chord error in the CAM system: Between 0.004 mm and 0.030 mm

- **Finishing with preference for high accuracy:**
  Use smaller values for the chord error and a matching low tolerance in Cycle 32. The data density must be high enough for the control to detect transitions and corners exactly. If a special cycle is available on your machine, use the finishing mode. In finishing mode the machine generally moves with low jerk values and low accelerations
  - Normal tolerance in Cycle 32: Between 0.002 mm and 0.006 mm
  - Normal chord error in the CAM system: Between 0.001 mm and 0.004 mm

- **Finishing with preference for high surface quality:**
  Use small values for the chord error and a matching larger tolerance in Cycle 32. The control is then able to better smooth the contour. If a special cycle is available on your machine, use the finishing mode. In finishing mode the machine generally moves with low jerk values and low accelerations
  - Normal tolerance in Cycle 32: Between 0.010 mm and 0.020 mm
  - Normal chord error in the CAM system: Approx. 0.005 mm

Further adaptations

Take the following points into account with CAM programming:

- For slow machining feed rates or contours with large radii, define the chord error to be only one-third to one-fifth of tolerance **T** in Cycle 32. Additionally, define the maximum permissible point spacing to be between 0.25 mm and 0.5 mm. The geometry error or model error should also be specified to be very small (max. 1 µm).

- Even at higher machining feed rates, point spacings of greater than 2.5 mm are not recommended for curved contour areas

- For straight contour elements, one NC point at the beginning of a line and one NC point at the end suffice. Avoid the output of intermediate positions

- In programs with five axes moving simultaneously, avoid large changes in the ratio of path lengths in linear and rotational blocks. Otherwise large reductions in the feed rate could result at the tool reference point (TCP)

- The feed-rate limitation for compensating movements (e.g. via **M128 F...**) should be used only in exceptional cases. The feed-rate limitation for compensating movements can cause large reductions in the feed rate at the tool reference point (TCP).

- NC programs for 5-axis simultaneous machining with spherical cutters should preferably be output for the center of the sphere. The NC data are then generally more consistent. Additionally, in Cycle 32 you can set a higher rotational axis tolerance **TA** (e.g. between 1° and 3°) for an even more constant feed-rate curve at the tool reference point (TCP).

- For NC programs for 5-axis simultaneous machining with toroid cutters or spherical cutters, where the NC output is for the south pole of the sphere, choose a lower rotary axis tolerance. 0.1° is a typical value. However, the maximum permissible contour damage is the decisive factor for the rotational axis tolerance. This contour damage depends on the possible tool tilting, tool radius and contact depth of the tool.
With 5-axis gear hobbing with an end mill you can calculate the maximum possible contour damage T directly from the cutter contact length L and permissible contour tolerance TA:
$T \sim K \times L \times TA$ $K = 0.0175$ [1/°]
Example: $L = 10$ mm, $TA = 0.1°$: $T = 0.0175$ mm

Possibilities for intervention on the control

Cycle 32 **TOLERANCE** is available for influencing the behavior of CAM programs directly on the control. Please note the information describing the functioning of Cycle 32. Also note the interactions with the chord error defined in the CAM system.

Further information: Cycle Programming User's Manual

> ⚙ Refer to your machine manual!
>
> Some machine tool builders provide an additional cycle for adapting the behavior of the machine to the respective machining operation, such as Cycle 332 Tuning. Cycle 332 can be used to modify filter settings, acceleration settings, and jerk settings.

Example

34 CYCL DEF 32.0 TOLERANCE
35 CYCL DEF 32.1 T0.05
36 CYCL DEF 32.2 HSC MODE:1 TA3

ADP motion control

> ⚙ This function must be enabled and adapted by the machine tool builder.

An insufficient quality of data in NC programs created on CAM systems frequently causes inferior surface quality of the milled workpieces. The **ADP** (Advanced Dynamic Prediction) feature expands the conventional look-ahead of the permissible maximum feed rate profile and optimizes the motion control of the feed axes during milling. This enables clean surfaces with short machining times to be cut, even with a strongly fluctuating distribution of points in adjacent tool paths. This significantly reduces or eliminates the reworking complexity.

These are the most important benefits of ADP:

- Symmetrical feed-rate behavior on forward and backward paths with bidirectional milling
- Uniform feed rate curves with adjacent cutter paths
- Improved reaction to negative effects (e.g. short, step-like stages, coarse chord tolerances, heavily rounded block end-point coordinates) in NC programs generated by CAM system
- Precise compliance to dynamic characteristics even in difficult conditions

6.10   Functions for program display

Overview

In the **Program Run Single Block** and **Program Run Full Sequence** operating modes, the control displays the following soft keys for displaying the NC program in pages:

Soft key	Functions
PAGE ↑	Go back one screen in the NC program
PAGE ↓	Go forward one screen in the NC - program
BEGIN ↑	Select start of program
END ↓	Select end of program

6.11 Automatic program start

Application

⚙️   Refer to your machine manual!
       The control must be specially prepared by the machine
       tool builder for use of the automatic program start
       function.

⚠️ DANGER

Caution: Danger for the operator!
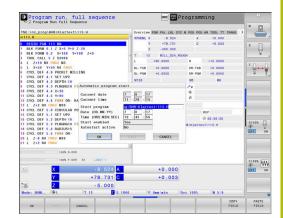
The **AUTOSTART** function automatically starts the machining
operation. Open machines with unsecured work envelopes pose
a huge danger for the machine operator.

▶ Use the **AUTOSTART** function exclusively on enclosed
  machines

In a Program Run operating mode, you can use the **AUTOSTART**
soft key to define a specific time at which the NC program that is
currently active in this operating mode is to be started:

AUTOSTART 🕐	▶ Display window for setting the starting time
	▶ **Time (h:min:sec):** Time of day at which the NC program is to be started
	▶ **Date (TT.MM.JJJJ):** Date at which the NC program is to be started
	▶ To activate the start, press the **OK**

6.12 Positioning w/ Manual Data Input operating mode

The **Positioning w/ Manual Data Input** operating mode is particularly convenient for simple machining operations or for pre-positioning the tool. It enables you to write a short NC program, depending on machine parameter **programInputMode** (no. 101201), in Klartext conversational programming or in ISO format and execute it immediately. The NC program is stored in the file $MDI.

You can use the following functions for example:

- Cycles
- Radius compensation
- Program section repetitions
- Q parameters

The additional status display can be activated in the **Positioning w/ Manual Data Input** operating mode.

NOTICE

Danger of collision!

Certain manual interactions cause the control to lose program information affecting the mode and thereby to lose the so-called contextual reference. After the loss of the contextual reference, unexpected and undesired movements can occur. There is a danger of collision during subsequent machining operations!

▶ Do not perform the following interactions:
  - Cursor movement to another NC block
  - The jump command **GOTO** to another NC block
  - Editing an NC block
  - Modifying Q parameter values with the **Q INFO** soft key
  - Switching the operating modes
▶ Restore the contextual reference via repetition of the required NC blocks

---

Positioning with manual data input (MDI)

▶ Select **Positioning w/ Manual Data Input** operating mode

▶ Program the desired available function

▶ Press the **NC Start** key

> The control executes the highlighted NC block.
  Further information: "Positioning w/ Manual Data Input operating mode", Page 306

---
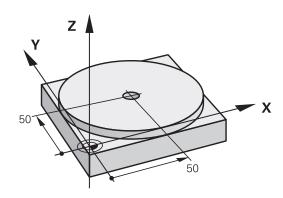
i Operating and programming notes:
  - The following functions are not available in the **Positioning w/ Manual Data Input** operating mode:
    - FK free contour programming
    - Program call
      - **PGM CALL**
      - **SEL PGM**
      - **CALL SELECTED PGM**
    - Programming graphics
    - Program-run graphics
  - Using the **SELECT BLOCK** and **CUT OUT BLOCK** soft keys etc. you can also conveniently and rapidly reuse program sections from other NC programs.
    Further information: User's Manuals for Conversational Programming and ISO Programming
  - You can control and modify Q parameters with the soft keys **Q PARAMETER LIST** and **Q INFO**.
    Further information: "Checking and changing Q parameters", Page 280

Example

A hole with a depth of 20 mm is to be drilled into a single workpiece. After clamping and aligning the workpiece and setting the preset, you can program and execute the drilling operation with a few lines of programming.

First you pre-position the tool above the workpiece with straight-line blocks and position with a safety clearance of 5 mm above the hole. Then drill the hole with Cycle **200 DRILLING**.

0 BEGIN PGM $MDI MM	
1 TOOL CALL 1 Z S2000	Call the tool: tool axis Z,
	spindle speed 2000 rpm
2 L Z+200 R0 FMAX	Retract the tool (F MAX = rapid traverse)
3 L X+50 Y+50 R0 FMAX M3	Move the tool at F MAX to a position above the hole, spindle on
4 CYCL DEF 200 DRILLING	Define the DRILLING cycle
Q200=5 ;SET-UP CLEARANCE	Set-up clearance of the tool above the hole
Q201=-20 ;DEPTH	Hole depth (algebraic sign=working direction)
Q206=250 ;FEED RATE FOR PLNGNG	Feed rate for drilling
Q202=5 ;PLUNGING DEPTH	Depth of each infeed before retraction
Q210=0 ;DWELL TIME AT TOP	Dwell time after every retraction in seconds
Q203=-10 ;SURFACE COORDINATE	Coordinate of the workpiece surface
Q204=20 ;2ND SET-UP CLEARANCE	Set-up clearance of the tool above the hole
Q211=0.2 ;DWELL TIME AT DEPTH	Dwell time in seconds at the hole bottom
Q395=0 ;DEPTH REFERENCE	Depth referenced to the tool tip or the cylindrical part of the tool
5 CYCL CALL	Call the DRILLING cycle
6 L Z+200 R0 FMAX M2	Retract the tool
7 END PGM $MDI MM	End of program

Example: Remove workpiece misalignment on a machine with a rotary table

▶ Use a 3-D touch probe to carry out a basic rotation
  Further information: "Compensating workpiece misalignment with 3-D touch probe ", Page 224

▶ Write down the rotation angle and cancel the basic rotation

▶ Select the operating mode: Press the **Positioning w/ Manual Data Input** key

▶ Select the rotary table axis, enter the rotation angle and feed rate you wrote down, e.g.**L C +2.561 F50**

▶ Conclude entry

▶ Press the **NC Start** button: The rotation of the table corrects the misalignment

Saving NC programs from $MDI

The $MDI file is intended for short NC programs that are only needed temporarily. Nevertheless, you can store an NC program, if necessary, by proceeding as follows:

▶ Operating mode: Press the **Programming** key

▶ To call the file manager, press the **PGM MGT** key.

▶ Move the highlight to the **$MDI** file

▶ To copy the file: Press the **COPY** soft key

DESTINATION FILE =

▶ Enter the name under which you want to save the current contents of the $MDI file, e.g. **Hole**

▶ Press the **OK** soft key.

▶ Exit the file manager: Press the **END** soft key

6.13 Entering miscellaneous functions M and STOP

Fundamentals

With the control's miscellaneous functions—also called M functions—you can affect:

- the program run, e.g. a program interruption
- the machine functions, such as switching spindle rotation and coolant supply on and off
- The path behavior of the tool

You can enter up to four M (miscellaneous) functions at the end of a positioning block or in a separate NC block. The control displays the following dialog question: **Miscellaneous function M ?**

You usually enter only the number of the miscellaneous function in the programming dialog. With some miscellaneous functions, the dialog is extended so that you can enter the required parameters for this function.

In the **Manual operation** and **Electronic handwheel** operating modes, the M functions are entered with the **M** soft key.

Effectiveness of miscellaneous functions

Please note that some M functions become effective at the start of a positioning block, and others at the end, regardless of their position in the NC block.

Miscellaneous functions come into effect in the NC block in which they are called.

Some miscellaneous functions are effective only in the NC block in which they are programmed. Unless the miscellaneous function is only effective blockwise, you must either cancel it in a subsequent NC block with a separate M function, or it is automatically canceled by the control at the end of the program.

> **ℹ**   If multiple functions were programmed in a single NC block, the execution sequence is as follows:
> - M functions taking effect at the start of the block are executed before those taking effect at the end of the block
> - If all M functions are effective at the start or end of the block, execution takes place in the sequence as programmed

Entering a miscellaneous function in a STOP block

If you program a **STOP** block, the program run or test run is interrupted at the block, e.g. for a tool inspection. You can also enter an M (miscellaneous) function in a **STOP** block:

| STOP | ▶ | To program an interruption of program run, press the **STOP** key |
| | ▶ | Enter a miscellaneous function **M** if required |

Example

```
87 STOP
```

6.14 Miscellaneous functions for program run inspection, spindle and coolant

Overview

⚙️ Refer to your machine manual!
The machine manufacturer can influence the behavior of the miscellaneous functions described below.
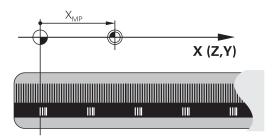
M	Effect	Effective at block	Start	End
M0	Program STOP Spindle STOP			■
M1	Optional program STOP Spindle STOP if necessary Coolant OFF if necessary (function defined by the machine tool builder)			■
M2	STOP program run Spindle STOP Coolant off Return jump to block 1 Clear status display Functional scope depends on machine parameter **resetAt** (no. 100901)			■
M3	Spindle ON clockwise		■	
M4	Spindle ON counterclockwise		■	
M5	Spindle STOP			■
M6	Tool change Spindle STOP Program STOP			■

ℹ️ Since this function varies depending on the machine tool builder, HEIDENHAIN recommends that you use the **TOOL CALL** function for tool changes.

M	Effect	Effective at block	Start	End
M8	Coolant ON		■	
M9	Coolant OFF			■
M13	Spindle ON clockwise Coolant ON		■	
M14	Spindle ON counterclockwise Coolant ON		■	
M30	Same as M2			■

6.15 Miscellaneous functions for coordinate entries

Programming machine-referenced coordinates: M91/M92

Scale datum

On the scale, a reference mark indicates the position of the scale datum.

Machine datum

The machine datum is required for the following tasks:

- Define the axis traverse limits (software limit switches)
- Approach machine-referenced positions (such as tool change positions)
- Set a workpiece preset

The distance in each axis from the scale datum to the machine datum is defined by the machine tool builder in a machine parameter.

Standard behavior

The control references the coordinates to the workpiece datum.

Further information: "Presetting without a 3-D touch probe", Page 203

Behavior with M91—Machine datum

If you want the coordinates in a positioning block to reference the machine datum, enter M91 into these NC blocks.

> **i** If you program incremental coordinates in an M91 block, enter them with respect to the last programmed M91 position. If no M91 position is programmed in the active NC block, then enter the coordinates with respect to the current tool position.

The coordinate values on the control's screen are referenced to the machine datum. Switch the display of coordinates in the status display to REF.

Further information: "Status displays", Page 68

Behavior with M92 – Additional machine reference point

> ⚙ Refer to your machine manual!
>
> In addition to the machine datum, the machine tool builder can also define an additional machine-based position as a machine reference point.
>
> For each axis, the machine tool builder defines the distance between the machine reference point and the machine datum.

If you want the coordinates in positioning blocks to be based on the machine preset, enter M92 into these NC blocks.

> ℹ Radius compensation remains the same in blocks that are programmed with **M91** or **M92**. The tool length will **not** be taken into account.

Effect

M91 and M92 are effective only in the blocks in which M91 and M92 have been programmed.
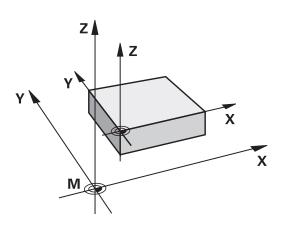
M91 and M92 take effect at the start of block.

Workpiece preset

If you want the coordinates to always reference to the machine datum, you can inhibit presetting for one or more axes.

If presetting is inhibited for all axes, the control no longer displays the **SET PRESET** soft key in the **Manual operation** mode.

The figure shows coordinate systems with the machine and workpiece datum.

M91/M92 in the Test Run mode

In order to be able to graphically simulate M91/M92 movements, you need to activate working space monitoring and display the workpiece blank referenced to the defined preset.

Further information: "Showing the workpiece blank in the working space ", Page 265

Moving to positions in a non-tilted coordinate system with a tilted working plane: M130

Standard behavior with a tilted working plane

The control references the coordinates in the positioning blocks to the tilted working plane coordinate system.

Behavior with M130

Despite an active tilted working plane, the control references the coordinates in straight line blocks to the non-tilted workpiece coordinate system.

The control then positions the tilted tool at the programmed coordinates of the non-tilted workpiece coordinate system.

NOTICE

Danger of collision!

The **M130** function is only active blockwise. The control executes the subsequent machining operations in the tilted working plane coordinate system again. Danger of collision during machining!

▶ Check the sequence and positions using a graphic simulation

---

ℹ Programming notes:
- The **M130** function is only allowed if the **Tilt the working plane** function is active.
- If the **M130** function is combined with a cycle call, the control will interrupt the execution with an error message.

Effect

M130 functions blockwise in straight-line blocks without tool radius compensation.

6.16 Miscellaneous functions for path behavior

Superimposing handwheel positioning during program run: M118

Standard behavior

⚙ Refer to your machine manual!
Your machine tool builder must have prepared the control for this function.

In the Program Run operating modes, the control moves the tool as defined in the NC program.

Behavior with M118

M118 permits manual corrections by handwheel during the program run. For this purpose, you program **M118** and enter an axis-specific value (linear or rotary axis).

ℹ The **M118** handwheel superimpositioning function, in combination with the **Dynamic Collision Monitoring (DCM)** function, can only be used at a standstill.

In order to use **M118** without restrictions, either deselect the **Dynamic Collision Monitoring (DCM)** function using the soft key from the menu or activate a kinematics model without collision objects (CMOs).

Input

If you enter **M118** in a positioning block, the control continues the dialog for this block by prompting you for the axis-specific values. Use the orange axis keys or the alphabetic keyboard for entering the coordinates.

Effect

To cancel handwheel positioning, program **M118** once again without coordinate input or end the NC program with **M30** / **M2**.

ℹ If the program aborts, handwheel positioning will also be canceled.

M118 becomes effective at the start of the block.

Example

You want to be able to use the handwheel during program run to move the tool in the working plane X/Y by ±1 mm and in the rotary axis B by ±5° from the programmed value:

```
L X+0  Y+38.5 RL  F125 M118  X1 Y1 B5
```

> **ℹ** When programmed in an NC program, **M118** is always effective in the machine coordinate system.
>
> If the Global Program Settings option (Option 44) is active, **Handradüberlagerung** is effective in the last selected coordinate system. The coordinate system active for Handradüberlagerung is shown on the **POS HR** tab of the additional status display.
>
> The **POS HR** tab also indicates whether the **Max. val.** have been defined via **M118** or via the Global Program Settings.
>
> **Further information:** "Handwheel superimp.", Page 359
>
> **Handradüberlagerung** is also effective in the **Positioning w/ Manual Data Input** operating mode!

Virtual tool axis (VT)(Option 44)

> **⚙** Refer to your machine manual!
>
> Your machine tool builder must have prepared the control for this function.

With the virtual tool axis, you can also traverse with the handwheel in the direction of a sloping tool on a machine with swivel heads. To traverse in a virtual tool axis direction, select the **VT** axis on the display of your handwheel.

Further information: "Traverse with electronic handwheels", Page 175

When using an HR 5xx handwheel, you can select the virtual axis directly with the orange **VI** axis key, if necessary.

In conjunction with the **M118** function, it is also possible to carry out handwheel superimpositioning in the currently active tool axis direction. For this purpose, program at least the spindle axis with its permitted range of traverse in the **M118** function (e.g. **M118 Z5**) and select the **VT** axis on the handwheel.

Deleting basic rotation: M143

Standard behavior

The basic rotation remains in effect until it is reset or is overwritten with a new value.

Behavior with M143

The control deletes a basic rotation from the NC program.

> ℹ️ The function **M143** is not permitted with mid-program startup.

Effect

M143 is effective only from the NC block in which it is programmed.

M143 becomes effective at the start of the block.

> ℹ️ **M143** clears the entries from the **SPA**, **SPB** and **SPC** columns in the preset table. When the corresponding line is reactivated, the basic rotation is **0** in all columns.

Automatically retracting the tool from the contour at an NC stop: M148

Standard behavior

In case of an NC stop, the control stops all traverse movements. The tool stops moving at the point of interruption.

Behavior with M148

> ⚙️ Refer to your machine manual!
>
> This function must be configured and enabled by your machine tool builder.
>
> In the **CfgLiftOff** (no. 201400) machine parameter, the machine tool builder defines the path the control is to traverse for a **LIFTOFF** command. You can also use the **CfgLiftOff** machine parameter to deactivate the function.

Set the **Y** parameter in the **LIFTOFF** column of the tool table for the active tool. The control then retracts the tool from the contour by 2 mm max. in the direction of the tool axis.

Further information: "Entering tool data into the table", Page 137

LIFTOFF takes effect in the following situations:

- An NC stop triggered by you
- An NC stop triggered by the software, e.g. if an error occurred in the drive system
- When a power interruption occurs

Effect

M148 remains in effect until deactivated with **M149**.

M148 becomes effective at the start of the block, **M149** at the end of the block.

7

Special Functions

7.1 Dynamic Collision Monitoring (option 40)

Function

> ⚙ Refer to your machine manual!
>
> The machine tool builder needs to adapt the **Dynamic Collision Monitoring (DCM)** function to the control.

The machine manufacturer can define any objects that will be monitored by the control during all machining operations. If two objects monitored for collision come within a defined distance of each other, the control generates an error message and terminates the movement.

The control can display the defined collision objects graphically in all Machine operating modes and in the **Test Run** operating mode.

Further information: "Graphic display of the collision objects", Page 322

The control also monitors the active tool for collision and displays the situation graphically. The control always assumes cylindrical tools. The control likewise monitors stepped tools according to their definition in the tool table.

The control takes into account the following definitions from the tool table:
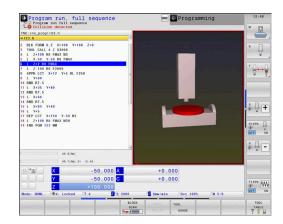
- Tool lengths
- Tool radii
- Tool dimensions
- Tool carrier kinematics

NOTICE

Danger of collision!

Even if **Dynamic Collision Monitoring (DCM)** is active, the control does not automatically monitor the workpiece for collisions, be it with the tool or with other machine components. There is a danger of collision during machining!

- ▶ Check the machining sequence using a graphic simulation
- ▶ Carefully test the NC program or program section in the **Program run, single block** operating mode

---

> **Generally valid constraints:**
> - The **Dynamic Collision Monitoring (DCM)** function helps to reduce the danger of collision. However, the control cannot consider all possible constellations during operation.
> - The control can only protect those machine components from collision that your machine tool builder has defined correctly with regard to dimensions, orientation and position.
> - The control can only monitor tools for which you have defined **positive tool radii** and **positive tool lengths** in the tool table.
> - When a touch probe cycle starts, the control no longer monitors the stylus length and ball-tip diameter so that you can also probe collision objects.
> - For certain tools (such as face milling cutters), the radius that would cause a collision can be greater than the value defined in the tool table.
> - **DL** and **DR** tool oversizes from the tool table are taken into account by the control. Tool oversizes from the **TOOL CALL** block are not accounted for.

Graphic display of the collision objects

Activate the graphic display of the collision objects as follows:

►   Select the desired operating mode

           ►   Press the **Screen layout** key

           ►   Select the desired screen layout

POSITION + MACHINE

PROGRAM + MACHINE

MACHINE

You can also use the soft keys to change the display of the collision objects.

Modify the graphic display of the collision objects as follows:

VIEW OPTIONS
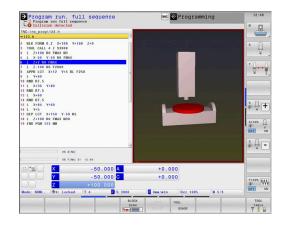
           ►   Press the **VIEW OPTIONS** soft key

           ►   Modify the graphic display of the collision objects
               Further information: "VIEW OPTIONS",
               Page 257

You can also use the mouse to change the display of the collision objects.

The following functions are available:

►   In order to rotate the model shown in three dimensions you hold the right mouse button down and move the mouse. If you simultaneously press the shift key, you can only rotate the model horizontally or vertically.

►   To shift the model shown: Hold the center mouse button or the wheel button down and move the mouse. If you simultaneously press the shift key, you can only shift the model horizontally or vertically.

►   To zoom in on a certain area: Mark a zoom area by holding the left mouse button down.

>   After you release the left mouse button, the control zooms in on the defined area.

►   To rapidly magnify or reduce any area: Rotate the mouse wheel backwards or forwards.

►   To return to the standard display: Press the shift key and simultaneously double-click with the right mouse key. The rotation angle is maintained if you only double-click with the right mouse key.

Collision monitoring in the manual operating modes

In the **Manual operation** and **Electronic handwheel** operating modes, the control stops the movement if two objects monitored for collision approach each other within a minimum distance. In this case, the control displays an error message naming the two objects causing collision.

> ⚙ Refer to your machine manual!
>
> The machine tool builder can define the minimum distance between two collision-monitored objects.

Before the collision warning, the control dynamically reduces the feed rate of the movements to ensure that the axes stop in good time before a collision.

If you have selected a screen layout in which the collision objects are on the right, then the control additionally marks the colliding objects in red.

> ℹ When a collision warning has been issued, machine movements via the axis direction keys or the handwheel are only possible if they increase the distance between the collision objects.
>
> With active collision monitoring and a simultaneous collision warning, no movements are permitted that reduce the distance or leave it unchanged.
>
> **Further information:** "Activating and deactivating collision monitoring", Page 327

> 📖 Note the general limitations of the **Dynamic Collision Monitoring (DCM)** function.
>
> **Further information:** "Function", Page 320

Collision monitoring in the Test Run operating mode

In the **Test Run** operating mode, you can perform collision monitoring of an NC program prior to machining. The control stops the simulation in case of a collision and displays an error message indicating the two objects that would cause the collision.

If you have selected a screen layout in which the collision objects are on the right, then the control additionally marks the colliding objects in red.

HEIDENHAIN recommends the use of Dynamic Collision Monitoring (DCM) in the **Test Run** operating mode only as an add-on to collision monitoring in the Machine operating mode.

Please note in Test Run operating mode

To obtain a simulation result that is similar to execution, the following aspects must match:

- Preset
- Basic rotation
- Offsets of each axis
- Tilting condition
- Activated kinematic model

The control automatically adopts the preset table, but you have to select the preset in the simulated NC program.

In addition, you can use the current machine status for the **Test Run** operating mode.

The current machine status includes the following:

- active machine kinematics
- active traverse ranges
- active machining modes
- active working spaces
- active preset

To load the current machine status, proceed as follows:

	▶	Press the **BLANK IN WORK SPACE** soft key

	▶	Press the **Load machine status** soft key
	>	The control simulates the current machine status.

In a simulation, the following aspects may differ from the actual machine or may not be available at all:

- The simulated tool change position may differ from the one in the machine operating mode.
- Changes in the kinematics may have a delayed effect in the simulation.
- PLC positioning movements are not displayed in the simulation.
- Global program settings and handwheel superimposition are not available.
- Pallet processing is not available in the simulation.
- Traverse range limits from the MOD function are not available

> 📖 Note the general limitations of the **Dynamic Collision Monitoring (DCM)** function.
> **Further information:** "Function", Page 320

Activating collision monitoring in the simulation

Proceed as follows to activate Dynamic Collision Monitoring in the **Test Run** operating mode:

▶ Select the **Test Run** operating mode

▶ Press the **Collision Monitoring ON** soft key

You can toggle collision monitoring only after the simulation has been stopped.

Collision monitoring in the Program Run operating modes

In the **Positioning w/ Manual Data Input**, **Program Run Single Block** and **Program run, full sequence** operating modes, the control stops the program run before an NC block in which two objects monitored for collision would approach each other within a distance of less than 5 mm is processed. In this case, the control displays an error message naming the two objects causing collision.

If you have selected a screen layout in which the collision objects are on the right, then the control additionally marks the colliding objects in red.

⚠ *NOTICE*

Danger of collision!

The machine tool builder has various options for configuring the **Dynamic Collision Monitoring (DCM)** function. Depending on the machine, the NC program will be continued without an error message despite a detected collision, but the tool will be stopped at the last position without collision. If the NC program allows for a new position without collision, the control resumes the machining operation and positions the tool at that position. This configuration of the **Dynamic Collision Monitoring (DCM)** function results in movements that are not defined in the program. **This process takes place no matter whether collision monitoring is active or inactive.** There is a danger of collision during these movements!

▶ Refer to your machine manual.
▶ Check the behavior at the machine.

ℹ   **Constraints with program run:**

■ For tapping with a floating tap holder, only the home position of the floating tap holder is taken into account by the **Dynamic Collision Monitoring (DCM)** function.

■ The **Handwheel superimp. M118** function can only be used in combination with the active **Dynamic Collision Monitoring (DCM)** function when program run has been stopped.

■ The **Dynamic Collision Monitoring (DCM)** function cannot be used in combination with the **M118** function and the additional **TCPM** or **M128** function.

■ If functions or cycles require multiple axes to be coupled (e.g. for eccentric turning), the control cannot perform collision monitoring.

■ If at least one axis operates with following error or is not referenced, the control cannot perform collision monitoring.

> 📖 Note the general limitations of the **Dynamic Collision Monitoring (DCM)** function.
> **Further information:** "Function", Page 320

Activating and deactivating collision monitoring

In some cases it is necessary to temporarily deactivate collision monitoring:

- To reduce the distance between two objects monitored for collision
- To prevent stops during program runs

NOTICE

Danger of collision!

If the **Dynamic Collision Monitoring (DCM)** function is inactive, the control does not perform any automatic collision checking. This means that movements that might cause collisions will not be prevented. There is a danger of collision during all movements!

- ▶ Make sure to activate collision monitoring whenever possible
- ▶ Make sure to always re-activate collision monitoring after a temporary deactivation
- ▶ With collision monitoring deactivated, carefully test the NC program or program section in the **Program run, single block** operating mode

---

You have the following possibilities:

- Permanently manually activating and deactivating collision monitoring
- Temporarily activating and deactivating collision monitoring in the NC program

Permanently manually activating and deactivating collision monitoring

▶ Operating mode: Press the **Manual operation** or **Electronic handwheel** key

▶ Shift the soft-key row if necessary

▶ Press the **COLLISION** soft key

▶ Select the operating modes for which the modification should apply:
  - **Program run**: **Positioning w/ Manual Data Input**, **Program run, single block** and **Program run, full sequence**
  - **Manual operation**: **Manual operation** and **Electronic handwheel**

▶ Press the **Go to** soft key

▶ Select the condition for which the selected operating modes should apply:
  - **Inactive**: Deactivate collision monitoring
  - **Active**: Activate collision monitoring
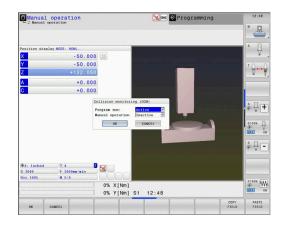
▶ Press the **Ok** soft key

Symbols

Icons in the status display show the condition of collision monitoring:

Icon	Function
	Collision monitoring active
	Collision monitoring is not available
	Collision monitoring is inactive

Activating and deactivating collision monitoring in the NC program

In some cases it is necessary to temporarily deactivate collision monitoring:

- To reduce the distance between two objects monitored for collision
- To prevent stops during program runs

NOTICE

Danger of collision!

If the **Dynamic Collision Monitoring (DCM)** function is inactive, the control does not perform any automatic collision checking. This means that movements that might cause collisions will not be prevented. There is a danger of collision during all movements!

▶ Make sure to activate collision monitoring whenever possible
▶ Make sure to always re-activate collision monitoring after a temporary deactivation
▶ With collision monitoring deactivated, carefully test the NC program or program section in the **Program run, single block** operating mode

---

Temporarily activating and deactivating collision monitoring via program control

▶ Open the NC program in **Programming** mode
▶ Place the cursor at the desired position, e.g. before Cycle 800 to enable eccentric turning

> **SPEC FCT**

▶ Press the **SPEC FCT** key

> **PROGRAM FUNCTIONS**

▶ Press the **PROGRAM FUNCTIONS** soft key

> **▷**

▶ Shift the soft-key row

> **FUNCTION DCM**

▶ Press the **FUNCTION DCM** soft key

> **FUNCTION DCM OFF**
>
> **FUNCTION DCM ON**

▶ Select the condition with the corresponding soft key:

- **FUNCTION DCM OFF**: This NC command temporarily deactivates collision monitoring. The deactivation is effective only until the end of the main program or until the next **FUNCTION DCM ON**. When another NC program is called, DCM is active again.
- **FUNCTION DCM ON**: This NC command cancels an existing **FUNCTION DCM OFF**.

ℹ️ The settings applied with the **FUNCTION DCM** function are only effective in the active NC program.

After terminating the program run or selecting a new NC program, the settings made for **Program run** and **Manual operation** with the **COLLISION** soft key become effective again.

Further information: "Activating and deactivating collision monitoring", Page 327

7.2 Adaptive Feed Control (AFC) (option 45)

Application

> ⚙️ This function must be enabled and adapted by the machine tool builder.
>
> Your machine tool builder may also specify whether the spindle power or any other value is used as input quantity by the control.
>
> If you have enabled the software option for turning (Option 50), you can use AFC in turning mode as well.

> ℹ️ Adaptive feed control is not intended for tools with diameters less than 5 mm. If the rated power consumption of the spindle is very high, the limit diameter of the tool may be larger.
>
> Do not work with adaptive feed control in operations in which the feed rate and spindle speed must be adapted to each other, such as tapping.
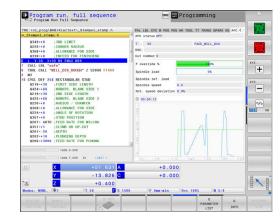
In adaptive feed control the control automatically controls the feed rate during NC program run as a function of the current spindle power. The spindle power required for each machining step is to be determined in a teach-in cut and saved by the control in a file belonging to the NC program. At the start of each machining step, usually when the spindle is switched on, the control controls the feed rate so that it remains within the limits that you have defined.

> ℹ️ If the cutting conditions do not change, you can define the spindle power consumption, which has been determined in a teach-in cut, as permanent tool-dependent reference power. Use the **AFC-LOAD** column in the tool table to do this. If you enter a value manually in this column, the control does not execute any more teach-in cuts.

This makes it possible to avoid negative effects on the tool, the workpiece, and the machine that might be caused by changing cutting conditions. Cutting conditions are changed particularly by:

- Tool wear
- Fluctuating cutting depths that occur especially with cast parts
- Fluctuating hardness caused by material flaws

Adaptive feed control (AFC) has the following advantages:

■ Optimization of machining time

By controlling the feed rate, the control tries to maintain the previously recorded maximum spindle power or the reference power specified in the tool table (**AFC-LOAD** column) during the entire machining time. It shortens the machining time by increasing the feed rate in machining zones with little material removal.

■ Tool monitoring

If the spindle power exceeds the recorded or specified maximum value (**AFC-LOAD** column of the tool table), the control decreases the feed rate until the reference spindle power is reached again. If the maximum spindle power is exceeded during machining and at the same time the feed rate falls below the minimum that you have defined, the control reacts by shutting down. This helps to prevent further damage after a tool breaks or is worn out.

■ Protection of the machine's mechanical elements

Timely feed rate reduction and shutdown responses help to avoid machine overload.

Defining basic AFC settings

In the **AFC.TAB** table, which must be saved in the **TNC:\table** directory, you enter the control settings with which the control performs the feed rate control.

The data in this table are default values that are copied into a file belonging to the respective NC program during a teach-in cut. The values act as the basis for feedback control.

> **i**   If you define a tool-specific feedback-control reference power using the **AFC-LOAD** column in the tool table, the control generates the associated file for the relevant NC program without a teach-in cut. The file is created shortly before feedback control becomes effective.

Enter the following data in the table:

Column	Function
NR	Consecutive line number in the table (has no further functions)
AFC	Name of the control setting. You enter this name in the **AFC** column of the tool table. It specifies the assignment of control parameters to the tool.
FMIN	Feed rate at which the control is to conduct a shutdown response. Enter the value in percent with respect to the programmed feed rate. Input range: 50 to 100 %
FMAX	Maximum feed rate in the material up to which the control can automatically increase the feed rate. Enter the value in percent of the programmed feed rate.
FIDL	Feed rate for traverse when the tool is not cutting. Enter the value in percent of the programmed feed rate.
FENT	Feed rate for traverse when the tool moves into or out of the material. Enter the value in percent with respect to the programmed feed rate. Maximum input value: 100 %
OVLD	Reaction that the control is to perform in case of overload: ■ **M**: Execution of a macro defined by the machine tool builder ■ **S**: Immediate NC stop ■ **F**: NC stop if the tool has been retracted ■ **E**: Just display an error message on the screen ■ **L**: Disable active tool ■ **-**: No overload reaction The control performs the selected overload reaction if, when feedback control is active, the maximum spindle power is exceeded for more than one second and at the same time the feed rate falls below the minimum you defined. Enter the desired function via the alphabetic keyboard. In conjunction with the cut-related tool wear monitoring function, the control will only evaluate the options **M**, **E**, and **L**! **Further information:** "Tool wear monitoring", Page 343
POUT	Spindle power at which the control is to detect that the tool moves out of the workpiece. Enter the value in percent of the learned reference load. Recommended input value: 8 %
SENS	Sensitivity (aggressiveness) of feedback control. A value between 50 and 200 can be entered. 50 is for slow control, 200 for a very aggressive control. An aggressive control reacts quickly and with strong changes to the values, but it tends to overshoot. Recommended value: 100
PLC	Value that the control is to transfer to the PLC at the beginning of a machining step. The machine manufacturer defines the function, so refer to your machine manual.

> **ⓘ** In the **AFC.TAB** table you can define as many control settings (lines) as desired.
>
> If there is no AFC.TAB table in the **TNC:\table** directory, the control uses a fixed control setting for the teach-in cut. If, alternatively, a tool-dependent reference power value exists, the control uses it immediately. HEIDENHAIN recommends to use the AFC.TAB table in order to ensure a safe and well-defined operation.

Proceed as follows to create the AFC.TAB file (only necessary if the file does not yet exist):

▶ Select the **Programming** operating mode

▶ To call the file manager, press the **PGM MGT** key

▶ Select the **TNC:** directory

▶ Create a new **AFC.TAB** file

▶ Press the **ENT** key

> The control displays a list with table formats.

▶ Select the **AFC.TAB** table format and confirm with the **ENT** key

> The control creates the table that contains the control settings.

Programming AFC

NOTICE
Caution: Danger to the tool and workpiece!
If you activate the **FUNCTION MODE TURN** machining mode, the control will clear the current **OVLD** values. This means that you need to program the machining mode before the tool call! If the programming sequence is not correct, no tool monitoring will take place, which might result in damage to the tool or workpiece!
▶ Program the **FUNCTION MODE TURN** machining mode before the tool call

Proceed as follows to program the AFC functions for starting and ending the teach in cut:

SPEC FCT	▶ Press the **SPEC FCT** key

PROGRAM FUNCTIONS	▶ Press the **PROGRAM FUNCTIONS** soft key

FUNCTION AFC	▶ Press the **FUNCTION AFC** soft key ▶ Select the function

The control provides several functions that enable you to start and stop AFC:

■ **FUNCTION AFC CTRL**: The **AFC CTRL** function activates feedback control mode starting with this NC block, even if the learning phase has not been completed yet.

■ **FUNCTION AFC CUT BEGIN TIME1 DIST2 LOAD3**: The control starts a sequence of cuts with active **AFC**. The changeover from the teach-in cut to feedback control mode begins as soon as the reference power has been determined in the teach-in phase, or once one of the **TIME**, **DIST** or **LOAD** conditions has been met.

  ■ With **TIME**, you define the maximum duration of the teach-in phase in seconds.

  ■ **DIST** defines the maximum distance for the teach-in cut.

  ■ With **LOAD**, you can set a reference load directly. If you enter a reference load > 100 %, the control automatically limits the value to 100 %.

■ **FUNCTION AFC CUT END**: The **AFC CUT END** function deactivates the AFC control.

> ⓘ The **TIME**, **DIST** and **LOAD** defaults are modally effective. They can be reset by entering **0**.

> ⓘ You can define a feedback-control reference power with the **AFC LOAD** tool table column and the **LOAD** input in the NC program. You can activate the **AFC LOAD** value via the tool call and the **LOAD** value with the **FUNCTION AFC CUT BEGIN** function.
>
> If you program both values, the control will use the value programmed in the NC program!

Opening the AFC table

With a teach-in cut, the control at first copies the basic settings for each machining step, as defined in the AFC.TAB table, to a file called **<name>.H.AFC.DEP**. **<name>** is the name of the NC program for which you have recorded the teach-in cut. In addition, the control measures the maximum spindle power consumed during the teach-in cut and saves this value in the table.

You can change the **<name>.H.AFC.DEP** file in **Programming** operating mode.

If necessary, you can even delete a machining step (entire line) there.

> ⓘ The **dependentFiles** machine parameter (no. 122101) must be set to **MANUAL** so that you can view the dependent files in the file manager.
>
> In order to edit the **<name>.H.AFC.DEP** file, you must first set the file manager so that all file types can be displayed (**SELECT TYPE** soft key).
>
> **Further information:** "Files", Page 81

Recording a teach-in cut

Requirements

Remember the following before you record a teach-in cut:

- If required, adapt the control settings in the AFC.TAB table
- Enter the desired control setting for all tools in the **AFC** column of the tool table TOOL.T
- Select the NC program for teach-in
- Activate the **AFC** function by soft key
  Further information: "Activating and deactivating AFC", Page 340

With a teach-in cut, the control at first copies the basic settings for each machining step, as defined in the AFC.TAB table, to a file called **<name>.H.AFC.DEP**.

<name> is the name of the NC program for which you have recorded the teach-in cut. In addition, the control measures the maximum spindle power consumed during the teach-in cut and saves this value to the table.

---

i   If you define a tool-specific feedback-control reference power using the **AFC-LOAD** column, the control stops executing teach-in cuts. The control immediately uses the specified value for feedback control. You specify the value for the tool-specific feedback-control reference power once in advance with a teach-in cut. If the cut conditions change, e.g. if the workpiece material changes, carry out a new teach-in cut.

---

i   You can define a reference power with the **AFC LOAD** tool table column and by entering a **LOAD** value in the NC program. You can activate the **AFC LOAD** value via the tool call and the **LOAD** value with the **FUNCTION AFC CUT BEGIN** function.

      If you program both values, the control will use the value programmed in the NC program!

Each row in the **<name>.H.AFC.DEP** file stands for a machining section, that you start with **FUNCTION AFC CUT BEGIN** and complete with **FUNCTION AFC CUT END**. You can edit all data of the **<name>.H.AFC.DEP** file for optimization purposes. If you have optimized the values from the AFC.TAB table, the control places a ***** in front of these control settings in the AFC column.

Further information: "Defining basic AFC settings", Page 332
Besides the data from the AFC.TAB table, the control also saves the following additional information in the **<name>.H.AFC.DEP** file:

Column	Function
NR	Number of the machining step
TOOL	Number or name of the tool with which the machining step was made (not editable)
IDX	Index of the tool with which the machining step was made (not editable)
N	Difference for tool call: ■ **0**: Tool was called by its tool number ■ **1**: Tool was called by its tool name
PREF	Reference load of the spindle. The control measures the value in percent with respect to the rated spindle power
ST	Status of the machining step: ■ **L**: In the next program run, a teach-in cut will be recorded for this machining step. The control will overwrite any existing values in this line ■ **C**: The teach-in cut was completed successfully. The next program run can be conducted with automatic feed control
AFC	Name of the control setting

> ⚙ Refer to your machine manual!
>
> You can teach any number of machining steps for a tool. Your machine tool builder will either make a function available for this, or will integrate this possibility in the functions for switching on the spindle.
>
> The functions for starting and ending a machining step are machine-dependent.

Operating notes:

- When you are performing a teach-in cut, the control shows the spindle reference power determined until this time in a pop-up window.

- You can reset the reference power in milling mode at any time by pressing the **PREF RESET** soft key. The control will then start a new teach-in phase.

- When you record a teach-in cut, the control internally sets the spindle override to 100%. Then you can no longer change the spindle speed.

- During the teach-in cut, you can influence the measured reference load by using the feed rate override to make any changes to the contouring feed rate.

- In a milling operation, you do not have to run the entire machining step in learning mode. If the cutting conditions do not change significantly, you can switch to control mode immediately. Press the **EXIT LEARNING** soft key, and the status changes from **L** to **C**.

- You can repeat a teach-in cut as often as desired. Manually change the status from **ST** back to **L**. If the programmed feed rate value is far too high and forces you to sharply decrease the feed rate override during the machining step, you will have to repeat the teach-in cut.

- If the determined reference load is greater than 2 %, the control changes the status from teach-in (**L**) to controlling (**C**). Adaptive feed control is not possible for smaller values.

- In **FUNCTION MODE TURN** machining mode, the minimum reference load is 5 %. Even if the control determines lower values, it will still use this minimum reference load. Thus, the overload limits (indicated as percentage values) are based on a minimum reference load of at least 5 %.

Selecting the AFC table

Proceed as follows to select and, if required, edit the
<name>.H.AFC.DEP file:

| | ▶ Select the **Program run, full sequence** operating mode |

| | ▶ Shift the soft-key row |

| AFC SETTINGS | ▶ Press the **AFC SETTINGS** soft key |
| | ▶ Make optimizations if required |

> **ℹ** Note that the **<name>.H.AFC.DEP** file is locked against editing as long as the NC program **<name>.H** is running.
>
> The control does not remove the editing lock until one of the following functions has been executed:
> - **M02**
> - **M30**
> - **END PGM**

You can also change the **<name>.H.AFC.DEP** file in **Programming** mode. If necessary, you can even delete a machining step (entire line) there.

> **ℹ** The **dependentFiles** machine parameter (no. 122101) must be set to **MANUAL** so that you can view the dependent files in the file manager.
>
> In order to edit the **<name>.H.AFC.DEP** file, you must first set the file manager so that all file types can be displayed (**SELECT TYPE** soft key).
>
> **Further information:** "Files", Page 81

Activating and deactivating AFC

> **NOTICE**
>
> **Caution: Danger to the tool and workpiece!**
>
> As soon as you deactivate the **AFC** function, the control immediately switches to the programmed machining feed rate. If the **AFC** function decreased the feed rate (e.g. due to wear) before it was deactivated, the control accelerates the feed rate up to the programmed value. This applies regardless of the method used for deactivating the function (soft key, feed rate potentiometer, etc.). This acceleration may result in damages to the tool or the workpiece!
>
> ▶ If it is imminent that the feed rate falls below the **FMIN** value, stop the machining operation (instead of deactivating the **AFC** function)
>
> ▶ Define the overload reaction for cases in which the feed rate falls below the **FMIN** value

▶ Press the **Program run, full sequence** key

▶ Shift the soft-key row
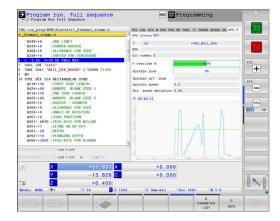
▶ To activate the adaptive feed control: Set the soft key to **ON**—the control displays the AFC symbol in the position display
  Further information: "Status displays", Page 68

▶ To deactivate the adaptive feed control: Set the soft key to **OFF**

i Operating notes:
- If adaptive feed control is active in **Control** mode, the control executes a shutdown response independent of the programmed overload reaction.
  - If, with the reference spindle load, the minimum feed factor is fallen below
  - If the programmed feed rate falls below the 30 % threshold
- If you do not explicitly deactivate the adaptive feed control using the soft key, this function remains active. The control remembers the setting of the soft key even if the power is interrupted.
- If the adaptive feed control is active in **Control** mode, the control internally sets the spindle override to 100 %. Then you can no longer change the spindle speed.
- If the adaptive feed control is active in **Control** mode, the control takes over the value from the feed rate override function.
  - Increasing the feed rate override has no influence on the control.
  - If you decrease the feed rate override by more than **10 %** with respect to the maximum setting, the control will switch off the adaptive feed control off. In this case, the control displays a window to inform you.
- In NC blocks containing **FMAX**, the adaptive feed control is **not active**.
- Mid-program startup is allowed during active feed control. The control takes the cutting number of the startup block in account.

The control shows various pieces of information in the additional status display when adaptive feed control is on.

Further information: "Additional status displays", Page 70

In addition, the control shows the ^{AFC}L or ^{AFC} icon in the position display.

Log file

The control stores various pieces of information for each machining step of a teach-in cut in the **<name>.H.AFC2.DEP** file. **<name>** is the name of the NC program for which you have recorded the teach-in cut. During adaptive control, the control updates the data and performs various evaluations. The following data will be saved in this table:

Column	Function
NR	Number of the machining step
TOOL	Number or name of the tool with which the machining step was made
IDX	Index of the tool with which the machining step was made
SNOM	Nominal spindle speed [rpm]
SDIFF	Maximum difference of the spindle speed in % of the nominal speed
CTIME	Machining time (tool in effect)
FAVG	Average feed rate (tool in effect)
FMIN	Smallest occurring feed factor. The control shows the value as a percentage of the programmed feed rate
PMAX	Maximum recorded spindle power during machining. The control shows the value as a percentage of the spindle's rated power.
PREF	Reference load of the spindle. The control shows the value as a percentage of the spindle's rated power.
OVLD	Overload reaction performed by the control:
	■ **M**: A macro defined by the machine tool builder has been run
	■ **S:** Immediate NC stop was conducted
	■ **F**: NC stop was conducted after the tool was retracted
	■ **E**: An error message was displayed
	■ **L**: The current tool was locked
	■ **-**: There was no overload reaction
BLOCK	Block number at which the machining step begins

> **ℹ** During regulation, the control indicates the current machining time as well as the resulting time saving in percent. The control enters the results of the evaluation between the key words **total** and **saved** in the last line of the log file. Where the time balance is positive, the percentage value is also positive.

Proceed as follows to select the **<name>.H.AFC2.DEP** file:

| | ▶ | Operating mode: Press the **Program run, full sequence** key |

⊞ (key icon)

| | ▶ | Shift the soft-key row |

◁ (key icon)

| AFC SETTINGS | ▶ | Press the AFC Settings soft key |

| TABLE EVALU- ATION | ▶ | Show the log file |

Tool wear monitoring

Activate cut-related tool wear monitoring by entering a value not equal to 0 in the **AFC-OVLD1** column in the tool table.

The shutdown response depends on the **AFC.TAB** column **OVLD**.

In conjunction with cut-related tool wear monitoring, the control only evaluates the options **M**, **E**, and **L** in the **OVLD** column. The following responses are possible:

- Pop-up window
- Lock current tool
- Insert replacement tool

> ℹ️ If the **AFC.TAB** columns **FMIN** and **FMAX** each have a value of 100 %, adaptive feed control is deactivated but cut-related tool wear monitoring remains active.

Further information: "Entering tool data into the table", Page 137 and Page 332

Tool load monitoring

Activate cut-related tool load monitoring (tool breakage control) by entering a value not equal to 0 in the **AFC-OVLD2** column in the tool table.

As shutdown response, the control always executes a machining stop and locks the momentary tool.

> ℹ️ If the **AFC.TAB** columns **FMIN** and **FMAX** each have a value of 100%, adaptive feed control is deactivated but cut-related tool load monitoring remains active.

Further information: "Entering tool data into the table", Page 137 and Page 332

7.3 Active Chatter Control ACC (Option 145)

Application

> ⚙ This function must be enabled and adapted by the machine tool builder.

Strong forces come into play during roughing (power milling). Depending on the tool spindle speed, the resonances in the machine tool and the chip volume (metal-removal rate during milling), the machine can sometimes begin to **chatter**. This chattering places heavy strain on the machine, and causes ugly marks on the workpiece surface. The tool, too, is subject to heavy and irregular wear from chattering. In extreme cases it can result in tool breakage.

In order to reduce a machine's tendency to chatter, HEIDENHAIN offers an effective control function known as Active Chatter Control (**ACC**). The use of this control function is particularly advantageous during heavy machining. ACC makes substantially higher metal removal rates possible. Depending on the type of machine, the metal-removal rate can often be increased by more than 25 %. You reduce the mechanical load on the machine and increase the life of your tools at the same time.

> ℹ ACC was developed especially for roughing and heavy machining and is particularly effective in this area. You need to conduct appropriate tests to see whether ACC will also be advantageous on your machine and with your tool.

Activating ACC

To activate ACC, proceed as follows:

- In the TOOL.T tool table, set column **ACC** to **Y**
- Specify the number of cutting edges for your tool in column **CUT** of the TOOL.T tool table
- Switch on the spindle
- The tooth meshing frequency must be between 20 and 150 Hz

If ACC is active, the control shows the ᴬᶜᶜ icon in the position display.

To activate/deactivate ACC temporarily for the machine mode, proceed as follows:

▶ Operating mode: Press the **Program run, full sequence**, **Program run, single block** or **Positioning w/ Manual Data Input** key

▶ Shift the soft-key row

▶ Activate ACC: Set the soft key to **ON**

> The control displays the ACC icon in the position display.
  Further information: "Status displays", Page 68

▶ To deactivate ACC: Set the soft key to **OFF**

7.4   Global Program Settings (option 44)

Application

> ⚙️ Refer to your machine manual!
>
> This function must be enabled and adapted by the machine tool builder.
>
> Your machine tool builder can also disable individual setting options within the **Global Program Settings** function.

The **Global Program Settings** function, which is mainly used in large-scale mold making, is available in the **Program run, full sequence** and **Program run, single block** operating modes, as well as in **Positioning with mdi**. They allow you to define various coordinate transformations and settings without having to edit the NC program. All of the settings apply globally and are superimposed on the selected NC program.

The **Global Program Settings** function and its settings remain active until they are reset. This also applies after the control has been restarted.

Further information: "Activating and deactivating a function", Page 348
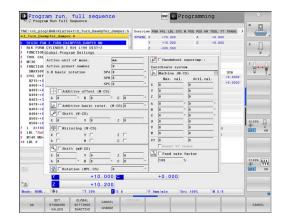
> ⚙️ Refer to your machine manual!
>
> Your machine tool builder specifies whether the **Global Program Settings** function also influences the manual cycles of the **Manual operation** mode.

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

Icon	Function	Description
![icon]	**Additive offset (M-CS)**	Page 351
![icon]	**Additive basic rotat. (W-CS)**	Page 352
![icon]	**Shift (W-CS)**	Page 353
![icon]	**Mirroring (W-CS)**	Page 355
![icon]	**Shift (mW-CS)**	Page 356
![icon]	**Rotation (WPL-CS)**	Page 358
![icon]	**Handwheel superimp.**	Page 359
![icon]	**Feed rate factor**	Page 362

i Operating notes:
- In the fillable form the control dims any axes that are not active on your machine.
- Entered values (e.g. shift values or values for **Handwheel superimp.**) are defined in the unit of measure (mm or inches) selected in the position display. Angles are always entered in degrees.
- Touch probe functions deactivate the **Global Program Settings** temporarily.
- If you want to use **Handwheel superimp.** while machining with **Dynamic Collision Monitoring (DCM)**, then the control must be in a stopped or interrupted state.
  Further information: "General status display", Page 68
  As an alternative, you can deactivate **Dynamic Collision Monitoring (DCM)**.
  Further information: "Activating and deactivating collision monitoring", Page 327

Activating and deactivating a function

The **Global Program Settings** function and its settings remain active until they are reset. This also applies after the control has been restarted.

As soon as any setting option of the **Global Program Settings** function is activated, the control shows the following symbol in the

position display:

Before machining you can use the form to activate or deactivate any of the setting options of the **Global Program Settings** function that have been enabled by the machine tool builder.

If you have interrupted the program run, you can also use the form to activate or deactivate **Handwheel superimp.** and the **Feed rate factor** during machining.
Further information: "Interrupting, stopping or aborting machining", Page 282
Once you restart the NC program, the control immediately applies the values you have defined. If necessary, the control approaches the new position via the menu for returning.
Further information: "Returning to the contour", Page 297

> **Refer to your machine manual!**
>
> The machine tool builder can provide functions with which you can set or reset **Handwheel superimp.** and the **Feed rate factor** under program control, e.g. M functions or manufacturer cycles.
>
> You can use Q-parameter functions to query the status of the **Global Program Settings** function.
> **Further information:** User's Manual for Conversational Programming or ISO Programming

Fillable form

Active setting options of the **Global Program Settings** function are highlighted white in the form. Inactive setting options remain dimmed.
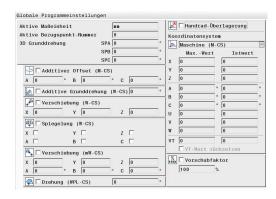
If more than one setting option for coordinate transformation (left half of the form) is active, the sequence of effect is shown using yellow numbers and arrows.
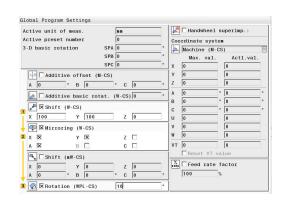
> The information area (top of left half of the form) and the setting options in the right half of the form are not considered for the sequence of effect, since they do not result in any coordinate transformations.

As soon as any setting option of the **Global Program Settings** function is activated, the control displays a warning message if an NC program is selected in the file manager.

Then you can simply acknowledge the message with **Ok** or call the form directly with **CHANGE DATA**.

Activating Global Program Settings

i	All changes must be confirmed with the **Ok** soft key. Otherwise the control discards the changes when closing the form, e.g. by pressing the **END** key.

GLOBAL
SETTINGS

▶ Press the **GLOBAL SETTINGS** soft key
> The control opens the form with the following elements:
   ■ Check boxes, e.g. for the setting options
   ■ Input fields for the entry of values
   ■ Pull-down menu of the coordinate systems for **Handwheel superimp.**
▶ Use form elements to activate a setting option
   Further information: "Using the form", Page 350

OK

▶ Press the **Ok** soft key
> The control applies the settings and closes the form

Deactivating Global Program Settings

i	All changes must be confirmed with the **Ok** soft key. Otherwise the control discards the changes when closing the form, e.g. by pressing the **END** key.

CHANGE
DATA

▶ After selecting the NC program, press the **CHANGE DATA** soft key

GLOBAL
SETTINGS

▶ Alternative: If the NC program is already open, press the **GLOBAL SETTINGS** soft key
> The control opens the form

GLOBAL
SETTINGS
INACTIVE

▶ Press the **GLOBAL SETTINGS INACTIVE** soft key in order to deactivate all setting options
▶ Alternative: Use form elements to deactivate a setting option
   Further information: "Using the form", Page 350

OK

▶ Press the **Ok** soft key
> The control applies the settings and closes the form

Using the form

Control	Function
	Jump to next setting option; if one is already activated, jump to next element
	Jump to previous setting option; if one is already activated, jump to previous element
Space	Activate or deactivate a selected check box (marked by a jump)
GOTO	Expand or collapse the pull-down menu
	Navigate in the pull-down menu
ENT / GOTO	Confirm the selection in the pull-down menu (and collapse the menu)
OK	Confirm the entries and close the form
SET STANDARD VALUES	Reset the entire form (exception: coordinate system selection for **Handwheel superimp.**)
GLOBAL SETTINGS INACTIVE	Deactivate all setting options without resetting other elements, such as values of input fields
ACTIVATE GLOBAL SETTINGS	Activate the most recently defined settings. After a restart of the control, you need to activate individual settings using the form elements.
CANCEL CHANGE	Discard all changes since the form was last called
CONFIRM VALUE	Apply actual values of **Handwheel superimp.** to the shifts. Prerequisite: The coordinate system for **Handwheel superimp.** and for **Displacement** concur

> **i** You can also easily navigate through the form with a mouse.

Information area

The form for the **Global Program Settings** function has an information area located in its upper left half. It contains the following:

- **Active unit of meas.**: Unit of measurement for entering values
  Further information: "Setting the unit of measure ",
  Page 429
- **Active preset number**: Preset management line
  Further information: "Activating a preset", Page 202
- **3-D basic rotation**: Spatial angle from preset management
  Further information: "General status display", Page 68 and
  Page 224

Active unit of meas.	mm
Active preset number	1
3-D basic rotation	SPA 0 °
	SPB 0 °
	SPC 0 °

Additive offset (M-CS)

> Refer to your machine manual!
>
> Your machine tool builder can also disable individual setting options within the **Global Program Settings** function.
>
> Axes that are not included in the kinematics description always appear dimmed and are therefore not editable!

☒ Additive offset (M-CS)
A 0 ° B 0 ° C 0 °

The **Additive offset (M-CS)** option of the **Global Program Settings** function provides a coordinate transformation in the M-CS machine coordinate system.
Further information: "Machine coordinate system M-CS",
Page 117

The additive offset of the **Global Program Settings** takes effect on an axis-by-axis basis. This value is added to the corresponding axis-specific offset from **Preset management**.
Further information: "Saving presets in the table", Page 196

> Refer to your machine manual!
>
> In machine parameter **presetToAlignAxis** (no. 300203) your machine tool builder specifies for each axis what effect an offset of a rotary axis has on the preset.
>
> - **True** (Default): Use offset for aligning the workpiece
> - **False**: Use offset for inclined-tool machining

NOTICE

Danger of collision!

The setting of the **presetToAlignAxis** (no. 300203) machine parameter determines whether a preset is taken into account in a rotary axis offset. There is a danger of collision during subsequent machining!

- ▶ Test the behavior at the machine
- ▶ If necessary, set a new preset after the offsets have been activated (mandatory for rotary axes on a table)

Control display

■ Both the additive offset of the **Global Program Settings** function and the offsets from **Preset management** affect the actual position display.

■ The general status display shows the following icons:

> **For offsets from Preset management, no icon is displayed!**

Additive offsets active (default icon of the **Global Program Settings** function)

■ The additive offset values are displayed on the **GS** tab of the additional status display. **Offsets from Preset management are exclusively displayed in Preset management!**

Example:

Increase traverse path:

■ Machine with AC fork head

■ Eccentric tool holder (outside the rotation center of the C axis)

■ **presetToAlignAxis** (no. 300203) machine parameter for the C axis is set to **FALSE**

■ Traverse path is increased by means of a 180° rotation of the C axis

■ Rotation is achieved by means of the **Additive offset (M-CS)** option

▶ Open the **Global Program Settings** function

▶ Activate the **Additive offset (M-CS)** option with C = 180°

▶ If necessary, add an **L C+0** positioning movement to the NC program

▶ Reselect the NC program

> The control considers the 180° rotation for all C axis positioning movements.

> The control takes the modified tool position into account.

> The position of the C axis does not affect the preset position. The preset remains unchanged!

Additive basic rotat. (W-CS)

⚙ Refer to your machine manual!
   Your machine tool builder can also disable individual setting options within the **Global Program Settings** function.

1 🔲 ☒ Additive basic rotat. (W-CS) 0   °

The **Additive basic rotat. (W-CS)** option of the **Global Program Settings** function provides a coordinate transformation in the W-CS workpiece coordinate system.
Further information: "Workpiece coordinate system W-CS", Page 120

The additive basic rotation of the **Global Program Settings** function is effective after the basic rotation or basic 3-D rotation and thus is based on this movement. This means that the value is not simply added to the SPC value of **Preset management**.
Further information: "Measuring 3-D basic rotation", Page 228 and Page 225

Control display

■ Like the basic rotation from **Preset management** (SPC column), the additive basic rotation of the **Global Program Settings** function does not affect the actual position display.

■ The general status display shows the following icons:

Active basic rotation from **Preset management**

Active basic 3-D rotation from **Preset management**

Additive basic rotation active (default icon of the **Global Program Settings** function)

■ The control displays the additive basic rotation values on the **GS** tab of the additional status display, the values from **Preset management** can be found on the **POS** tab.

Example:

Rotate the CAM output by –90°:

■ CAM output for gantry-type milling machine with a large range of traverse of the Y axis

■ Available machining center with a limited range of traverse of the Y axis (X axis has the required range of traverse)

■ The workpiece blank is clamped with a 90° rotation (long side parallel to the X axis)

■ Thus, the NC program must be rotated by 90° (algebraic sign depends on the preset position)

■ The 90° rotation is compensated by means of the **Additive basic rotat. (W-CS)** option

▶ Open the **Global Program Settings** function

▶ Activate the **Additive basic rotat. (W-CS)** option, specifying 90°

▶ Reselect the NC program

> The control considers the 90° rotation for all axis positioning movements.

Shift (W-CS)

⚙ Refer to your machine manual!

Your machine tool builder can also disable individual setting options within the **Global Program Settings** function.

The **Shift (W-CS)** option of the **Global Program Settings** function provides a coordinate transformation in the W-CS workpiece coordinate system.
Further information: "Workpiece coordinate system W-CS", Page 120

The **Shift (W-CS)** option of the **Global Program Settings** function takes effect on an axis-by-axis basis. The value is added to the shift that takes place **before** the working plane is tilted as defined in the NC program (e.g. Cycle 7 **DATUM SHIFT**).

Control display

- Unlike a datum shift in the NC program, the **Shift (W-CS)** option of the **Global Program Settings** function affects the actual position display.
- The general status display shows the following icons:

> **No icon is displayed for shifts defined in the NC program!**

 Shift (W-CS) function active (default icon of the **Global Program Settings** function)

- The **Shift (W-CS)** values are displayed on the **GS** tab of the additional status display, the values from the NC program on the **TRANS** tab.

Example:

Determining the workpiece position using the handwheel:

- Rework required on a tilted surface
- Workpiece clamped and roughly oriented
- Basic rotation and preset in the plane have been measured
- Z coordinate must be defined with the handwheel due to the presence of a free-form surface
- ▶ Open the **Global Program Settings** function
- ▶ Activate **Handwheel superimp.** with the **Workpiece (W-CS)** coordinate system
- ▶ Determine (touch off) the workpiece surface using the handwheel
- ▶ Transfer the determined value to **Shift (W-CS)** by pressing the **CONFIRM VALUE** soft key
- ▶ Continue the NC program
- ▶ Activate **Handwheel superimp.** with the **Workpiece (WPL-CS)** coordinate system
- ▶ Determine the workpiece surface using the handwheel (touch off for fine adjustment)
- ▶ Continue the NC program
- > The control takes the **Shift (W-CS)** into account.
- > The control uses the current values from **Handwheel superimp.** in the **Workpiece (WPL-CS)** coordinate system.

Mirroring (W-CS)

⚙ Refer to your machine manual!

Your machine tool builder can also disable individual setting options within the **Global Program Settings** function.

Axes that are not included in the kinematics description always appear dimmed and are therefore not editable!

The **Mirroring (W-CS)** option of the **Global Program Settings** function provides a coordinate transformation in the W-CS workpiece coordinate system.
Further information: "Workpiece coordinate system W-CS", Page 120

The **Mirroring (W-CS)** option of the **Global Program Settings** function takes effect on an axis-by-axis basis. The value is added to the mirroring that takes place **before** the working plane is tilted as defined in the NC program (e.g. Cycle 8 **MIRROR IMAGE**).

ℹ If **PLANE** functions or the **TCPM** function are used with spatial angles, the rotary axes are mirrored accordingly along with the mirrored principal axes. This always creates the same constellation, regardless of whether the rotary axes were marked in the form.

With **PLANE AXIAL**, the mirroring of rotary axes is irrelevant.

For the **TCPM** function with axis angles, all axes to be mirrored must be marked explicitly in the form.

Control display

- Like a shift in the NC program, the **Mirroring (W-CS)** of the **Global Program Settings** function has no effect on the actual position display.
- The general status display shows the following icons:

  Mirroring in the NC program active

  Mirroring (W-CS) function active (default icon of the **Global Program Settings** function)

- The **Mirroring (W-CS)** values are displayed on the **GS** tab of the additional status display, the values from the NC program on the **TRANS** tab.

Example:

Mirroring the CAM output:

- CAM output for right mirror cover
- The workpiece datum is centered on the workpiece blank
- NC program set to the center of the ball-nose cutter and **TCPM** function with spatial angles
- The left mirror cover is to be machined (X axis mirroring)
- ▶ Open the **Global Program Settings** function
- ▶ Activate **Mirroring (W-CS)** with marked X
- ▶ Run the NC program
- ＞ The control takes the **Mirroring (W-CS)** value for the X axis and the required rotary axes into account.

Shift (mW-CS)

⚙ Refer to your machine manual!
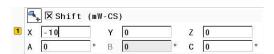
Your machine tool builder can also disable individual setting options within the **Global Program Settings** function.

The Shift (mW-CS) option of the **Global Program Settings** function provides a coordinate transformation in the mW-CS (modified workpiece coordinate system).

The W-CS workpiece coordinate system is modified with active **Shift (W-CS)** or active **Mirroring (W-CS)**. Without this preceding coordinate transformation, the Shift (mW-CS) option would be effective directly in the W-CS workpiece coordinate system and would thus be identical to **Shift (W-CS)**.
Further information: "Workpiece coordinate system W-CS", Page 120

The Shift (mW-CS) option of the **Global Program Settings** function takes effect on an axis-by-axis basis. The value is added to the shift that takes place **before** the working plane is tilted as defined in the NC program (e.g. Cycle 7 **DATUM SHIFT**), the same way as it is done for an active **Shift (W-CS)**.

Control display

- Unlike a datum shift in the NC program, the Shift (mW-CS) option of the **Global Program Settings** function has an effect on the actual position display.
- The general status display shows the following icons:

> **No icon is displayed for shifts defined in the NC program!**

> Shift (mW-CS) function active (default icon of the **Global Program Settings** function)

- The Shift (mW-CS) values are displayed on the **GS** tab of the additional status display, the values from the NC program on the **TRANS** tab.

⚙ Refer to your machine manual!

The shift (mW-CS) of the rotary axes to be taken into account is defined by your machine tool builder in the **presetToAlignAxis** (no. 300203) machine parameter.

- **True** (Default): Use offset for aligning the workpiece
- **False**: Use offset for inclined-tool machining

NOTICE

Danger of collision!

Machine parameter **presetToAlignAxis** (no. 300203) defines whether the shift (mW-CS) of a rotary axis is taken into account. There is a danger of collision during subsequent machining!

▶ Test the behavior at the machine

Example:

Mirroring the CAM output:
- CAM output for right mirror cover
- The workpiece datum is located in the left front corner of the workpiece blank.
- NC program set to the center of the ball-nose cutter and **TCPM** function with spatial angles
- The left mirror cover is to be machined (X axis mirroring)
- ▶ Open the **Global Program Settings** function
- ▶ Activate **Mirroring (W-CS)** with marked X
- ▶ Enter and activate Shift (mW-CS) to shift the workpiece datum in the mirrored coordinate system
- ▶ Run the NC program
- > The control takes the **Mirroring (W-CS)** value for the X axis and the required rotary axes into account.
- > The control takes the modified position of the workpiece datum into account.

Rotation (WPL-CS)

⚙	Refer to your machine manual!
	Your machine tool builder can also disable individual setting options within the **Global Program Settings** function.

The **Rotation (WPL-CS)** option of the **Global Program Settings** function provides a coordinate transformation in the WPL-CS working plane coordinate system.
Further information: "Working plane coordinate system WPL-CS", Page 122

The **Rotation (WPL-CS)** option of the **Global Program Settings** function is effective **after** tilting the working plane and thus is based on this movement. The value is added to the rotation defined in the NC program (e.g. Cycle 10 **ROTATION**).

Control display
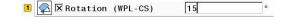
- Like a rotation in the NC program, the **Rotation (WPL-CS)** option of the **Global Program Settings** function has no effect on the actual position display.
- The general status display shows the following icons:

  No icon is displayed for rotations in the NC program!

  Rotation (WPL-CS) function active (default icon of the **Global Program Settings** function)

- The **Rotation (WPL-CS)** values are displayed on the **GS** tab of the additional status display, the values from the NC program on the **TRANS** tab.

Handwheel superimp.

⚙   Refer to your machine manual!

Your machine tool builder can also disable individual setting options within the **Global Program Settings** function.

The **Handwheel superimp.** option of the **Global Program Settings** function enables the axes to be moved in superimposition during the execution of an NC program. The coordinate system effective for **Handwheel superimp.** can be selected from the **Coordinate system** pull-down menu.

Icon	Function
⊞	**Handwheel superimp.** is effective in the machine coordinate system (M-CS) **Further information:** "Machine coordinate system M-CS", Page 117
⊞	**Handwheel superimp.** is effective in the workpiece coordinate system (W-CS) **Further information:** "Workpiece coordinate system W-CS", Page 120
⊞	**Handwheel superimp.** is effective in the modified workpiece coordinate system (mW-CS) **Further information:** "Shift (mW-CS)", Page 356
⊞	**Handwheel superimp.** is effective in the working plane coordinate system (WPL-CS) **Further information:** "Working plane coordinate system WPL-CS", Page 122

ℹ️   If no coordinate system transformations were activated using either the NC program or the **Global Program Settings** function, **Handwheel superimp.** is effective in the same way in all coordinate systems.

NOTICE
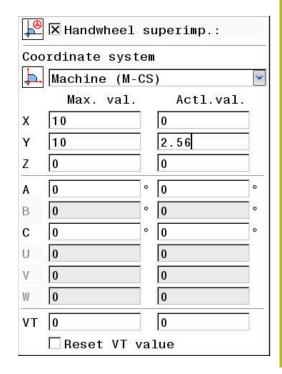
Danger of collision!

The coordinate system selected in the pull-down menu also takes effect on **Handwheel superimp.** with **M118**, even if the **Global Program Settings** function is not active. There is risk of collision during the execution of **Handwheel superimp.** and during the subsequent machining operations!

▶ Before exiting the form, always make sure to explicitly select the **Machine Coordinate System (M-CS)**.

▶ Test the behavior at the machine

By entering values in the **Max. val.** column, you define which axes may be moved using the handwheel and the maximum path by which they may be moved. Since the input value applies to both directions (both positive and negative), the maximum path is double the input value.

In the **Actl.val.** column, the control displays the path traversed using the handwheel for each axis.

The **Actl.val.** column can also be edited manually. However, if you enter a value that exceeds the current **Max. val.**, it will be impossible to activate the value. The wrong value will be highlighted in red. Moreover, the control displays a warning message and prevents you from closing the form.

If the **Actl.val.** column contains a value when you activate the function, the control will use the menu for returning to move to the new position.
Further information: "Returning to the contour", Page 297

---

i   Using the **CONFIRM VALUE** soft key, you can transfer the axis-specific values from the **Actl.val.** column to the shift values defined in the **Global Program Settings** function. This transfer is only possible for the principal axes. In addition, the coordinate systems must match.
Further information: "Shift (W-CS)", Page 353 and Page 356

When the values are applied, the control resets the input fields of the **Actl.val.** column.

If you apply the values more than once, the control will sum up the shift values.

NOTICE

Danger of collision!

If both methods for setting **Handwheel superimp.**, i.e. with **M118** or by using the **Global Program Settings** function, are effective at the same time, the definitions influence each other, depending on their sequence of activation. There is risk of collision during the execution of **Handwheel superimp.** and during the subsequent machining operations!

▶ If possible, only use one method for **Handwheel superimp.**
▶ Preferably use the **Handwheel superimp.** option of the **Global Program Settings** function
▶ Test the behavior at the machine

HEIDENHAIN does not recommend using both methods of setting **Handwheel superimp.** at the same time. If it is impossible to remove **M118** from the NC program, you should at least activate **Handwheel superimp.** of the **Global Program Settings** function prior to selecting the program. This ensures that the control uses the **Global Program Settings** function rather than **M118**.

ℹ Operating notes:
- In the fillable form the control dims any axes that are not active on your machine.
- Entered values (e.g. shift values or values for **Handwheel superimp.**) are defined in the unit of measure (mm or inches) selected in the position display. Angles are always entered in degrees.
- If you want to use **Handwheel superimp.** while machining with **Dynamic Collision Monitoring (DCM)**, then the control must be in a stopped or interrupted state.
  Further information: "General status display", Page 68
  As an alternative, you can deactivate **Dynamic Collision Monitoring (DCM)**.
  Further information: "Activating and deactivating collision monitoring", Page 327

Control display
- Both methods for **Handwheel superimp.** have an effect on the actual position display.
- The general status display shows the following icons:

  No icon is displayed for the M118 function!

  Handwheel superimp. active (default icon of the **Global Program Settings**)
- The control displays both values possible for **Handwheel superimp.** on the **POS HR** tab of the additional status display.
- The control indicates on the **POS HR** tab whether the Max. val. have been defined via M118 or the Global Program Settings.

Virtual tool axis (VT)

You can use **Handwheel superimp.** also in the currently active tool axis direction. Here, the current tool axis is the virtual axis **VT**, which does not correspond to the original tool axis direction **Z**. For activating this function, the **VT** (**V**irtual **T**ool axis) line is available in the form.

Values traversed with the handwheel in a virtual axis remain active in the default setting (check box unchecked) even after a tool change. The **Reset VT value** function allows you to change this behavior.

The virtual axis **VT** is frequently needed for machining operations with inclined tools, e.g. for manufacturing oblique holes without using a tilted working plane.

ℹ **Handwheel superimp.** in virtual axis direction **VT** requires neither one of the **PLANE** functions nor the **TCPM** function.

Display of the virtual tool axis (VT)

Handwheel superimp. must be activated with a virtual tool axis (**VT**) > 0 for the control to display values.

The virtual tool axis values (**VT**) are displayed on the **POS HR** tab of the additional status display.

If you have defined the virtual tool axis in the **axisDisplay** (no. 100810) machine parameter, the control also displays the **VT** axis in the position display.

Feed rate factor

⚙ Refer to your machine manual!

Your machine tool builder can also disable individual setting options within the **Global Program Settings** function.

With the **Feed rate factor** option, the **Global Program Settings** function allows you to modify the current machining feed rate. The input corresponds to a percentage. Input range: 1 % to 1000 %

ℹ The current machining feed rate is a combination of the programmed feed rate and the current position of the feed rate potentiometer.

ℹ The **Feed rate factor** option in the **Global Program Settings** has no influence on a programmed rapid traverse (**FMAX**).

All feed rates can jointly be limited using the feed rate limit (**F MAX** soft key). The **Feed rate factor** in the **Global Program Settings** function has no influence on the limited feed rate!
Further information: "Feed rate limit F MAX", Page 187

Control display

■ The general status display shows the following icons and information:

Ovr Result of the feed rate potentiometer setting

No icon and no value are displayed for the feed rate limit (F MAX soft key)!

Feed rate factor active (default icon of the **Global Program Settings** function)

F Result of all modifications = current feed rate

■ The control displays the value of the **feed rate factor** on the **GS** tab of the additional status display.

7.5   Defining a counter

Application

| ⚙ | Refer to your machine manual!
Your machine manufacturer enables this function. |

The **FUNCTION COUNT** function allows you to control a simple counter from within the NC program. For example, this function allows you to count the number of manufactured workpieces.

Proceed as follows for the definition:

| SPEC FCT | ▶ | Show the soft key row with special functions |

| PROGRAM FUNCTIONS | ▶ | Press the **PROGRAM FUNCTIONS** soft key |

| FUNCTION COUNT | ▶ | Press the **FUNCTION COUNT** soft key |

NOTICE

Caution: Data may be lost!

Only one counter can be managed by the control. If you execute an NC program that resets the counter, any counter progress of another NC program will be deleted.

▶ Please check prior to machining whether a counter is active.
▶ If necessary, note down the counter value and enter it again via the MOD menu after execution.

| ℹ | You can use Cycle 225 to engrave the current counter value into the workpiece.
Further information: Cycle Programming User's Manual |

Effect in the Test Run operating mode
You can simulate the counter in the **Test Run** operating mode. Only the count you have defined directly in the NC program is effective. The count in the MOD menu remains unaffected.

Effect in the Program Run Single Block and Program Run Full Sequence operating modes
The count from the MOD menu is only effective in the **Program Run Single Block** and **Program Run Full Sequence** operating modes.
The count is retained even after a restart of the control.

Defining FUNCTION COUNT

The **FUNCTION COUNT** function provides the following possibilities:

Soft key	Meaning
FUNCTION COUNT INC	Increase count by 1
FUNCTION COUNT RESET	Reset counter
FUNCTION COUNT TARGET	Set the nominal count (target value) to the desired value Input value: 0–9999
FUNCTION COUNT SET	Set the counter to the desired value Input value: 0–9999
FUNCTION COUNT ADD	Increment the counter by the desired value Input value: 0–9999
FUNCTION COUNT REPEAT	Repeat the NC program starting from this label if more parts are to be machined.

Example

5 FUNCTION COUNT RESET	Reset the counter value
6 FUNCTION COUNT TARGET10	Enter the target number of parts to be machined
7 LBL 11	Enter the jump label
8 L ...	Machining
51 FUNCTION COUNT INC	Increment the counter value
52 FUNCTION COUNT REPEAT LBL 11	Repeat the machining operations if more parts are to be machined.
53 M30	
54 END PGM	

8

Pallets

8.1 Pallet management

Application

⚙ Refer to your machine manual!

Pallet table management is a machine-dependent function. The standard functional range is described below.

Pallet tables (**.p**) are mainly used in machining centers with pallet changers. The pallet tables call the different pallets (PAL), fixtures (FIX) optionally, and the associated NC programs (PGM). The pallet tables activate all defined presets and datum tables.

Without a pallet changer you can use pallet tables to process NC programs with different presets in sequence with just one press of **NC Start**.

ℹ The file name of a pallet table must always begin with a letter.

Columns of the pallet table

The machine tool builder defines a pallet table prototype that opens automatically when you create a pallet table.

The prototype can include the following columns:

Column	Meaning	Field type
NR	The control creates the entry automatically. The entry is required for the input field **Line number** of the **BLOCK SCAN** function.	Mandatory field
TYPE	The control differentiates between the following entries ■ **PAL** Pallet ■ **FIX** Fixture ■ **PGM** NC program Select the entries using the **ENT** key and the arrow keys or by soft key.	Mandatory field
NAME	File name The machine tool builder specifies the names for pallets and fixtures, if applicable, whereas you define program names. You must specify the complete path if the NC program is not saved in the directory of the pallet table.	Mandatory field
DATUM	Datum You must specify the complete path if the datum table is not saved in the directory of the pallet table. You activate datums from a datum table in the NC program using Cycle 7.	Optional field This entry is only required if a datum table is used.
PRESET	Workpiece preset Enter the preset number of the workpiece.	Optional field

Column	Meaning	Field type
LOCATION	Location of the pallet The entry **MA** indicates that there is a pallet or fixture in the working space of the machine and can be machined. Press the **ENT** key to enter **MA**. Press the **NO ENT** key to remove the entry and thus suppress machining.	Optional field If the column exists, the entry is mandatory.
LOCK	Line locked Using an ***** you can exclude the line of the pallet table from processing. Press the **ENT** key to identify the line with the entry *****. Press the **NO ENT** key to cancel the lock. You can lock the execution for individual NC programs, fixtures or entire pallets. Unlocked lines (e.g. PGM) in a locked pallet are also not executed.	Optional field
PALPRES	Number of the pallet preset	Optional field This entry is only required if pallet presets are used.
W-STATUS	Execution status	Optional field This entry is only required for tool-oriented machining.
METHOD	Machining method	Optional field This entry is only required for tool-oriented machining.
CTID	ID for mid-program startup	Optional field This entry is only required for tool-oriented machining.
SP-X, SP-Y, SP-Z	Clearance height in the linear axes X, Y, and Z	Optional field
SP-A, SP-B, SP-C	Clearance height in the rotary axes A, B, and C	Optional field
SP-U, SP-V, SP-W	Clearance height in the parallel axes U, V, and W	Optional field
DOC	Comment	Optional field

> ℹ️ You can remove the **LOCATION** column if you are only using pallet tables in which the control is to machine all lines.
>
> **Further information:** "Inserting or deleting columns", Page 369

Editing a pallet table

When you create a new pallet table, it is empty at first. Using the soft keys, you can insert and edit lines.

Soft key	Editing function
BEGIN	Select the table start
END	Select the table end
PAGE	Select the previous page in the table
PAGE	Select the next page in the table
INSERT LINE	Insert as last line in the table
DELETE LINE	Delete the last line in the table
APPEND N LINES AT END	Add several lines at end of table
COPY FIELD	Copy the current value
PASTE FIELD	Insert the copied value
BEGIN LINE	Select beginning of line
END LINE	Select end of line
FIND	Find text or value
SORT/ HIDE COLUMNS	Sort or hide table columns
EDIT CURRENT FIELD	Edit the current field
SORT	Sort by column contents
MORE FUNCTIONS	Miscellaneous functions, e.g. saving
SELECT	Open file path selection

Selecting a pallet table

Proceed as follows to select a pallet table or create a new pallet table:

⬦	▶ Switch to the **Programming** mode or a program run mode
PGM MGT	▶ Press the **PGM MGT** key

If no pallet tables are shown:

SELECT TYPE	▶ Press the **SELECT TYPE** soft key
	▶ Press the **SHOW ALL** soft key
	▶ Select a pallet table with the arrow keys, or enter a name for a new pallet table (**.p**)
ENT	▶ Press the **ENT** key

> **i** You can select either a list view or form view using the **Screen Layout** key.

Inserting or deleting columns

> **i** This function is not enabled until the code number **555343** is entered.

Depending on the configuration, a newly created pallet table may not contain all columns. For tool-oriented machining, for example, you need columns that you have to insert first.

Proceed as follows to insert a column in an empty pallet table:
▶ Open the pallet table

MORE FUNCTIONS	▶ Press the **MORE FUNCTIONS** soft key
EDIT FORMAT	▶ Press the **EDIT FORMAT** soft key
	> The control opens a pop-up window displaying the available columns
	▶ Using the arrow keys, select the desired column.
INSERT COLUMN	▶ Press the **INSERT COLUMN** soft key
ENT	▶ Press the **ENT** key

You can remove the column with the **DELETE COLUMN** soft key.

Executing a pallet table

> ℹ️ A machine parameter defines whether the control is to execute the pallet table blockwise or continuously.

Proceed as follows to execute a pallet table:

⬦ ▶ Switch to **Program run, full sequence** or **Program run, single block** operating mode

[PGM MGT] ▶ Press the **PGM MGT** key

If no pallet tables are shown:

[SELECT TYPE] ▶ Press the **SELECT TYPE** soft key
▶ Press the **SHOW ALL** soft key
▶ Select a pallet table with the arrow keys

[ENT] ▶ Press the **ENT** key

[◯] ▶ Select the screen layout, if necessary

[NC Start] ▶ Execute with the **NC Start** key

Proceed as follows to check the NC program content before execution:

▶ Select pallet table
▶ With the arrow keys, select the NC program you would like to check

[OPEN THE PROGRAM] ▶ Press the **OPEN THE PROGRAM** soft key
> The control displays the selected NC program on the screen.

[↓] ▶ Scroll through the NC program with the arrow keys
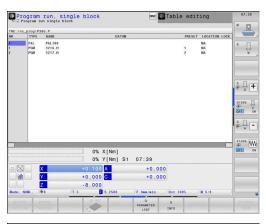
[END PGM → PAL] ▶ Press the **END PGM PAL** soft key
> The control returns to the pallet table.

> ℹ️ A machine parameter defines how the control is to react after an error.

Screen layout when executing the pallet table

If you want to see the NC program content and the content
of the pallet table at the same time, select the screen layout
PALLET + PROGRAM. During execution, the control then shows NC
program blocks to the left and the pallet to the right.

Editing pallet tables

If the pallet table is active in the **Program run, full sequence**
or **Program run, single block** operating mode, the soft keys for
modifying the table in the **Programming** operating mode are
inactive.

You can modify this table with the **EDIT PALLET** soft key in the
Program run, single block or **Program run, full sequence**
operating mode.

Block scan in a pallet table

With the pallet management you can also use the **BLOCK SCAN**
function in conjunction with pallet tables.

If you interrupt the execution of a pallet table, the control always
suggests the previously selected NC block of the interrupted NC
program for the **BLOCK SCAN** function.

Further information: "Block scan in pallet programs", Page 296

8.2   Pallet preset management

Fundamentals

⚙   Refer to your machine manual!

This function must be enabled and adapted by the machine tool builder.

Changes to the pallet preset table must always be made in agreement with your machine tool builder!

The pallet preset table is available to you in addition to the workpiece preset table (**preset.pr**). The workpiece presets refer to an activated pallet preset.

The control shows the active pallet preset in the status display on the PAL tab.

Application

Pallet presets are an easy way to compensate e.g. mechanical differences between individual pallets.

You can also completely align the coordinate system to the pallet by e.g. positioning the pallet preset in the center of a clamping tower.

Using pallet presets

If you want to use pallet presets, insert the **PALPRES** column in the pallet table.

Enter the preset number from the pallet preset table into this column. Usually, you always want to change the pallet preset when you insert a new pallet, i.e. in the PAL type lines of the pallet table.

NOTICE

Danger of collision!

Despite a basic rotation due to the active pallet preset, the control does not show a symbol in the status display. There is a danger of collision during all subsequent axis movements!

▶   If necessary, check the active pallet preset in the **PAL** tab

▶   Check the traverse movements of the machine

▶   Use pallet presets only in conjunction with pallets

8.3 Tool-oriented machining

Fundamentals of tool-oriented machining

Application

> ⚙ Refer to your machine manual!
>
> Tool-oriented machining is a machine-dependent function. The standard functional range is described below.

Tool-oriented machining allows you to machine several workpieces together even on a machine without pallet changer, which reduces tool-change times.

Limitation

> ### *NOTICE*
>
> **Danger of collision!**
>
> Not all pallet tables and NC programs are suitable for tool-oriented machining. With tool-oriented machining, the control no longer executes the NC programs continuously, but divides them at the tool calls. The division of the NC programs allows functions that were not reset to be effective across programs (machine states). This leads to a danger of collision during machining!
>
> ▶ Consider the stated limitations
> ▶ Adapt pallet tables and NC programs to the tool-oriented machining
>   ▪ Reprogram the program information after each tool in every NC program (e.g. **M3** or **M4**).
>   ▪ Reset special functions and miscellaneous functions before each tool in every NC program (e. g. **Tilt the working plane** or **M138**)
> ▶ Carefully test the pallet table and associated NC programs in the **Program run, single block** operating mode

The following functions are not permitted:
▪ FUNCTION TCPM, M128
▪ M144
▪ M101
▪ M118
▪ Changing the pallet preset

The following functions require special attention, particularly for mid-program startup:
▪ Changing the machine statuses with a miscellaneous function (e.g. M13)
▪ Writing to the configuration (e.g. WRITE KINEMATICS)
▪ Traverse range switchover
▪ Cycle 32 Tolerance
▪ Cycle 800
▪ Tilting the working plane

Pallet table columns for tool-oriented machining

Unless the machine tool builder has made a different configuration, you need the following additional columns for tool-oriented machining:

Column	Meaning
W-STATUS	The machining status defines the machining progress. Enter BLANK for an unmachined (raw) workpiece. The control changes this entry automatically during machining.
	The control differentiates between the following entries
	■ BLANK / no entry: Workpiece blank, requires machining
	■ INCOMPLETE: Partly machined, requires further machining
	■ ENDED: Machined completely, no further machining required
	■ EMPTY: Empty space, no machining required
	■ SKIP: Skip machining
METHOD	Indicates the machining method
	Tool-oriented machining is also possible with a combination of pallet fixtures, but not for multiple pallets.
	The control differentiates between the following entries
	■ WPO: Workpiece oriented (standard)
	■ TO: Tool oriented (first workpiece)
	■ CTO: Tool oriented (further workpieces)
CTID	The control automatically generates the ID number for mid-program startup with block scan.
	If you delete or change the entry, mid-program startup is no longer possible.
SP-X, SP-Y, SP-Z, SP-A, SP-B, SP-C, SP-U, SP-V, SP-W	The entry for the clearance height in the existing axes is optional.
	You can enter safety positions for the axes. The control only approaches these positions if the machine tool builder processes them in the NC macros.

Sequence of tool-oriented machining

Requirements

Requirements for tool-oriented machining:

- The machine manufacturer must define a tool-change macro for tool-oriented machining
- The tool-oriented machining methods TO and CTO have to be defined in the pallet table
- The NC programs are using the same tools to at least some extent
- The W-STATUS of the NC programs permits further machining

Sequence

1   The entry TO or CTO tells the control that the tool-oriented machining is valid beyond these lines of the pallet table

2   The control executes the NC program with the entry TO up to the TOOL CALL

3   The W-STATUS changes from BLANK to INCOMPLETE and the control enters a value into the CTID field

4   The control executes all other NC programs with the entry CTO up to the TOOL CALL

5   The control uses the next tool for the following machining steps if one of the following situations applies:
   - The next line in the table contains the entry PAL
   - The next line in the table contains the entry TO or WPO
   - There are lines in the table that do not yet contain the entry ENDED or EMPTY

6   The control updates the entry in the CTID field with each machining operation

7   If all table lines of the group contain the entry ENDED, the control processes the next few lines in the pallet table

Resetting the machining status

If you want to start machining again, change the W-STATUS to BLANK or remove the previous input.

If you change the status in the PAL line, all FIX and PGM lines below this line are automatically changed, too.

Mid-program startup with block scan

You can also return to a pallet table after an interruption. The control can show the line and the NC block at which the interruption occurred.

The block scan in the pallet table is tool oriented.

After a block scan, the control can resume tool-oriented machining if the tool-oriented machining method TO and CTO is defined in the following lines.

Keep the following in mind for mid-program startup

- The entry in the CTID field remains there for two weeks. After this time, mid-programs startup is no longer possible.
- Do not change or delete the entry in the CTID field.
- The data from the CTID field become invalid after a software update.
- The control saves the preset numbers for mid-program startup. If you change this preset, machining is shifted, too.
- Mid-program startup is no longer possible after editing an NC program within tool-oriented machining.

The following functions require special attention, particularly for mid-program startup:

- Changing the machine statuses with a miscellaneous function (e.g. M13)
- Writing to the configuration (e.g. WRITE KINEMATICS)
- Traverse range switchover
- Cycle 32 Tolerance
- Cycle 800
- Tilting the working plane

8.4 Batch Process Manager (option 154)

Application

> ⚙ Refer to your machine manual!
>
> Your machine tool builder configures and enables the
> **Batch Process Manager** function.

The **Batch Process Manager** enables you to plan production orders
on a machine tool.

You save the planned NC programs in a job list. You use the **Batch
Process Manager** to open the job list.

The following information is displayed:

- Whether the NC program is free of errors
- Run time of the NC programs
- Availability of the tools
- Times at which manual interventions in the machine are
  required

> ℹ The tool usage test function has to be enabled and
> switched on to ensure you get all information!
> **Further information:** "Tool usage test", Page 149

Fundamentals

The **Batch Process Manager** is available in the following operating
modes:
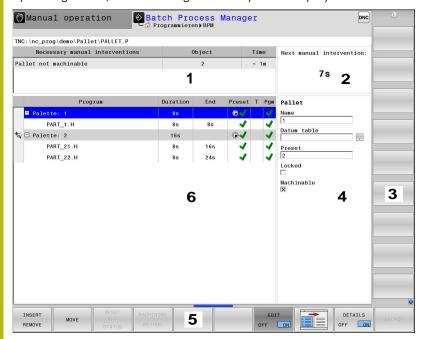
- **Programming**
- **Program run, single block**
- **Program run, full sequence**

In the **Programming** operating mode, you can create and edit the
job list.

The job list is executed in the **Program run, single block** and
Program run, full sequence operating modes. Changes are only
possible to a limited extent.

Screen display

When you open the **Batch Process Manager** in the **Programming** operating mode, the following screen layout is displayed:

1	Displays all required manual interventions
2	Displays the next manual intervention
3	Displays the current soft keys provided by the machine tool builder if available
4	Shows the editable entries in the line highlighted in blue
5	Displays the current soft keys
6	Displays the job list

Columns of the job list

Column	Meaning
No column name	Status of the **Pallet**, **Clamping**, or **Program**
Program	Name or path of the **Pallet**, **Clamping**, or **Program**
Duration	Run time in seconds This column is only shown if you have a 19-inch screen.
End	End of the run time ■ Time in **Programming** operating mode ■ Actual time in **Program run, single block** and **Program run, full sequence** operating modes
Preset	Status of the workpiece preset
T	Status of the inserted tools
Pgm	Status of the NC program
Sts	Machining status

The status of the **Pallet**, **Clamping**, and **Program** is shown by means of icons in the first column.

The icons have the following meanings:

Icon	Meaning
🔒	The **Pallet**, **Clamping**, or **Program** is locked
	The **Pallet** or **Clamping** is not enabled for machining
➡	This line is currently being processed in **Program run, single block** or **Program run, full sequence** and cannot be edited
➡	The program was interrupted manually in this line

In the **Program** column, the machining method is indicated by icons.

The icons have the following meanings:

Icon	Meaning
No icon	Workpiece-oriented machining
⌐	Tool-oriented machining ■ Start ■ End

The status is indicated by icons in the **Preset**, **T** and **Pgm** columns.

The icons have the following meanings:

Icon	Meaning
✔	Test completed
✔	Test completed Program simulation with active **Dynamic Collision Monitoring (DCM)** (Option 40)
✘	Test failed, e.g. because of expired tool life, danger of collision
⧗	Test not yet completed
?	Incorrect program structure, e.g.: pallet does not contain subordinate programs
⊕	Workpiece preset is defined
⚠	Check input You can either assign a workpiece preset to the pallet or to all subordinate NC programs.

Operating notes:
- In **Programming** operating mode, the **T** column is always empty, because the control first checks the status in the **Program run, single block** and **Program run, full sequence** operating modes
- If the tool usage test function is not enabled or switched on on your machine, no icon is shown in the **Pgm** column
Further information: "Tool usage test", Page 149

In the **Sts** columns, the machining status is indicated by icons.

The icons have the following meanings:

Icon	Meaning
	Workpiece blank, requires machining
	Partly machined, requires further machining
	Machined completely, no further machining required
	Skip machining

Operating notes:
- The machining status is automatically adjusted during machining
- The **Sts** column is only shown in the **Batch Process Manager** if the pallet table contains the **W STATUS** column

    Further information: "Tool-oriented machining", Page 373

Opening the Batch Process Manager

Refer to your machine manual!

In machine parameter **standardEditor** (no. 102902), your machine tool builder specifies the standard editor used by the control.

Programming operating mode

If the control does not open the pallet table (.p) in the Batch Process Manager as a job list, proceed as follows:

▶ Select the desired job list

▷	▶ Shift the soft-key row
MORE FUNCTIONS	▶ Press the **MORE FUNCTIONS** soft key
SELECT EDITOR	▶ Press the **SELECT EDITOR** soft key > The control opens the **Select editor** pop-up window.
↓	▶ Select **BPM-EDITOR**
ENT	▶ Confirm your entry with the **ENT** key
OK	▶ Alternative: Press the **OK** soft key > The control opens the job list in the **Batch Process Manager**.

Program run, single block and Program run, full sequence operating modes

If the control does not open the pallet table (.p) in the Batch Process Manager as a job list, proceed as follows:

◯	▶ Press the **Screen layout** key
BPM	▶ Press the **BPM** key > The control opens the job list in the **Batch Process Manager**.

Soft keys

The following soft keys are available:

> ⚙ Refer to your machine manual!
> The machine tool builder can configure his own soft keys.

Soft key	Function
DETAILS OFF ON	Collapse or expand tree structure
EDIT OFF ON	Edit opened job list
INSERT REMOVE	Shows the soft keys **INSERT BEFORE**, **INSERT AFTER** and **REMOVE**
MOVE	Move line
TAG	Select line

Soft key	Function
CANCEL THE MARKING	Cancel marking
INSERT BEFORE	Insert a new **Pallet**, **Clamping** or **Program** before the cursor position
INSERT AFTER	Insert a new **Pallet**, **Clamping**, or **Program** after the cursor position
REMOVE	Delete line or block
	Switch active windows
SELECT	Select possible entries from a pop-up window
RESET THE STATUS	Reset the machining status to workpiece blank
MACHINING METHOD	Select workpiece-oriented or tool-oriented machining
KOLLISIONS PRÜFUNG	Perform collision checking (Option 40) **Further information:** "Dynamic Collision Monitoring (option 40)", Page 320
KOLLISIONS PRÜFUNG ABBRECHEN	Abort collision checking (Option 40)
EINGRIFFE AUS EIN	Collapse or expand necessary manual interventions
TOOL MANAGEMENT	Open Extended Tool Management
INTERNAL STOP	Interrupt machining

ℹ Operating notes:

- The **TOOL MANAGEMENT**, **COLLISION CHECKING**, **ABORT COLLISION MONITORING** , and **INTERNAL STOP** soft keys are only available in the **Program run, single block** and **Program run, full sequence** operating modes.
- If the pallet table contains the **W STATUS** column, the **RESET THE STATUS** soft key is available.
- If the pallet table contains the **W STATUS**, **METHOD** and **CTID** columns, the **MACHINING METHOD** soft key is available.
  Further information: "Tool-oriented machining", Page 373

Creating a job list

You can only create a new job list in the file manager.

> ⓘ The file name of a job list must always begin with a letter.

⬥ ▶ Press the **Programming** key

PGM MGT ▶ Press the **PGM MGT** key
> The control opens the file manager.

NEW FILE ▶ Press the **NEW FILE** soft key

▶ Enter the file name with extension (**.p**)

ENT ▶ Confirm with the **ENT** key
> The control opens an empty job list in the **Batch Process Manager**.

INSERT REMOVE ▶ Press the **INSERT REMOVE** soft key

INSERT AFTER ▶ Press the **INSERT AFTER** soft key
> The control displays the various types on the right-hand side.

▶ Select the desired type
■ **Pallet**
■ **Clamping**
■ **Program**

> The control inserts an empty line in the job list.
> The control shows the selected type on the right-hand side.

▶ Define the entries
■ **Name**: Enter the name directly or select one by means of the pop-up window, if there is one
■ **Datum table**: Enter the datum directly, where applicable, or select one by means of the pop-up window
■ **Preset**: Enter the workpiece preset directly, where applicable
■ **Locked**: The selected line is excluded from machining
■ **Machinable**: The selected line is enabled for machining

ENT ▶ Confirm your entries by pressing the **ENT** key.

▶ Repeat the steps if required

EDIT OFF ON ▶ Press the **EDIT** soft key

Editing a job list

You can edit a job list in the **Programming**, **Program run, single block**, and **Program run, full sequence** operating modes.

ℹ️   Operating notes:

- If a job list is selected in the **Program run, single block** or **Program run, full sequence** operating mode, it is not possible to edit the job list in the **Programming** operating mode.
- The possibilities of changing a job list during machining are limited, because the control defines a protected area.
- NC programs in the protected area are shown in light gray.
- If you edit the job list, the Collision checking completed ✔️ status is reset to Check completed ✔️.

Proceed as follows to edit a line in the job list in the **Batch Process Manager**:

▶   Open the desired job list

[EDIT OFF ON]   ▶   Press the **EDIT** soft key

[↓]   ▶   Place the cursor on the desired line, e.g. **Pallet**

> The control displays the selected line in blue.

> The control displays the editable entries on the right-hand side.

[icon]   ▶   Press the **CHANGE WINDOW** soft key if required

> The control switches the active window.

▶   The following entries can be changed:

     ■   **Name**

     ■   **Datum table**

     ■   **Preset**

     ■   **Locked**

     ■   **Machinable**

[ENT]   ▶   Confirm the edited entries by pressing the **ENT** key.

> The control adopts the changes.

[EDIT OFF ON]   ▶   Press the **EDIT** soft key

Proceed as follows to move a line in the job list in the **Batch Process Manager**:

▶ Open the desired job list

▶ Press the **EDIT** soft key

▶ Place the cursor on the desired line, e.g. **Program**

> The control displays the selected line in blue.

▶ Press the **MOVE** soft key

▶ Press the **TAG** soft key

> The control highlights the line in which the cursor is positioned.

▶ Place the cursor on the desired position.

> When the cursor is placed at a suitable position, the control shows the **INSERT BEFORE** and **INSERT AFTER** soft keys.

▶ Press the **INSERT BEFORE** soft key

> The control inserts the line at the new position.

▶ Press the **GO BACK** soft key

▶ Press the **EDIT** soft key

9

Turning

9.1 Turning operations on milling machines (option 50)

Introduction

Special types of milling machines allow performing both milling and drilling operations. A workpiece can thus be machined completely on one machine without rechucking, even if complex milling and turning applications are required.

Turning is a machining operation during which the workpiece rotates and thus performs the cutting movement. A fixed tool carries out infeed and feed movements.

Turning applications, depending on machining direction and task, are subdivided into various production processes, e.g.

- Longitudinal turning
- Face turning
- Recess turning
- Thread cutting

> 📖 The control offers you several cycles for each of the various production processes.
>
> **Further information:** Cycle Programming User's Manual

On the control you can simply switch between milling and turning mode within the NC program. In turning mode, the rotary table serves as lathe spindle, whereas the milling spindle with the tool is fixed. This enables rotationally symmetric contours to be created. The preset must be in the center of the lathe spindle for this.

When managing turning tools, other geometric descriptions than those for milling or drilling tools are required. To be able to execute tool radius compensation, for example, you have to define the tool radius. The control provides special tool management for turning tools to support this definition process.

Further information: "Tool data", Page 400
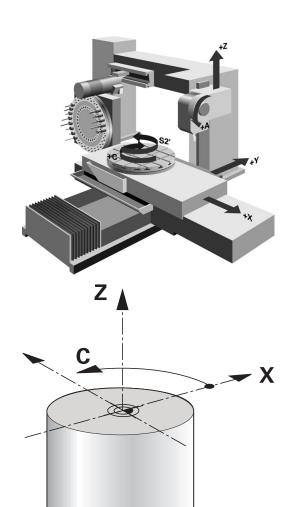
Different cycles are available for machining. These can also be used with additional swivel axes.

Further information: User's Manual Conversational Programming

Coordinate plane of turning operations

The assignment of the axes with turning is defined so that the X coordinates describe the diameter of the workpiece and the Z coordinates the longitudinal positions.

Programming is thus always done in the ZX coordinate plane. The machine axes to be used for the required movements depend on the respective machine kinematics and are determined by the machine manufacturer. This makes NC programs with turning functions largely exchangeable and independent of the machine model.

Tool radius compensation TRC

The tip of a lathe tool has a certain radius (**RS**). When machining tapers, chamfers and radii, this results in distortions on the contour because the programmed traverse paths refer to the theoretical tool tip S. TRC prevents the resulting deviations.

In the turning cycles the control automatically carries out tool radius compensation. In specific traversing blocks and within programmed contours, activate TRC with **RL** or **RR**.

The control checks the cutting geometry with the point angle **P-ANGLE** and the setting angle **T-ANGLE**. Contour elements in the cycle are processed by the control only as far as this is possible with the specific tool.

The control displays a warning when residual material is left behind due to the angle of the secondary cutting edges. You can suppress the warning with the machine parameter **suppressResMatlWar** (no. 201010).

> ℹ️ Programming notes:
>
> - The direction of the radius compensation is not clear when the tool-tip position (**TO=2, 4, 6, 8**) is neutral. In this case, TRC is only possible within fixed machining cycles.
>
>   The control can also run tool tip radius compensation during inclined processing.
>
>   Active miscellaneous functions limit the possibilities here:
>
>   - With **M128** tool-tip radius compensation is possible only in combination with machining cycles
>
>   - **M144** or **FUNCTION TCPM** with **REFPNT TIP-CENTER** also allows tool-tip radius compensation with all positioning blocks, e.g. with **RL/RR**

Theoretical tool tip

The theoretical tool tip is effective in the tool coordinate system. When the tool is inclined, the position of the tool tip rotates with the tool.

Virtual tool tip

Use **FUNCTION TCPM** with the selection **REFPNT TIP-CENTER** to activate the virtual tool tip. Correct tool data are the prerequisite for calculating the virtual tool tip.

The virtual tool tip is effective in the workpiece coordinate system. When the tool is inclined, the virtual tool tip remains unchanged as long as the tool orientation **TO** is the same. The control automatically switches the status display **TO** and thus also the virtual too tip if the tool leaves the angle range valid for **TO 1**, for example.

The virtual tool tip enables you to perform inclined paraxial longitudinal and transverse machining operations with high contour accuracy even without radius compensation.

9.2 Basic functions (option 50)

Switching between milling and turning mode

⚙ Refer to your machine manual!

The machine tool builder configures and enables turning and switchover of the machining modes.

To switch between milling and turning operations, you must switch to the specific mode.

You can switch these operating modes with the NC functions **FUNCTION MODE TURN** and **FUNCTION MODE MILL**.

If turning mode is active, the control displays a corresponding icon in the status display.

Icon	Mode
	Turning mode active: **FUNCTION MODE TURN**
No icon	Milling mode active: **FUNCTION MODE MILL**

When the operating modes are switched, the control executes a macro that defines the machine-specific settings for the specific operating mode. With the NC functions **FUNCTION MODE TURN** and **FUNCTION MODE MILL** you can activate a machine kinematic model that the machine tool builder has defined and saved in the macro.

NOTICE

Caution: Significant property damage!

Very high physical forces are generated during turning, for example by high rotational speeds and heavy or unbalanced workpieces. Incorrect machining parameters, neglected unbalances or improper fixtures lead to an increased risk of accidents during machining!

▶ Clamp the workpiece in the spindle center
▶ Clamp workpiece securely
▶ Program low spindle speeds (increase as required)
▶ Limit the spindle speed (increase as required)
▶ Eliminate unbalance (calibrate)

---

i Programming notes:

- If the **Tilt working plane** or **TCPM** functions are active, you cannot switch the operating mode.
- In turning mode, no coordinate conversion cycles are permitted except for the datum shift.
- The orientation of the tool spindle (spindle angle) depends on the machining direction. The tool tip is aligned to the center of the turning spindle for outside machining. For inside machining, the tool points away from the center of the turning spindle.
- The direction of spindle rotation must be adapted when the machining direction (outside/inside machining) is changed.
- During turning, the cutting edge and the center of the turning spindle must be at the same level. During turning, the tool therefore has to be pre-positioned to the Y coordinate of the turning-spindle center.
- By means of M138, you can select the rotary axes for M128 and TCPM.

i Operating notes:

- The preset must be in the center of the turning spindle in turning mode.
- In turning mode, diameter values are displayed on the X axis position display. The control then shows an additional diameter symbol.
- In turning mode, the spindle potentiometer is active for the turning spindle (rotary table).
- In turning mode you can use all manual touch probe cycles, except the **Probe corner** and **Probe plane** cycles. In turning mode, the measured values of the X axis equal diameter values.
- You can also use the smartSelect function to define the turning functions.

Specifying the machining mode

| SPEC FCT | ▶ Show the soft-key row with special functions |

| FUNCTION MODE | ▶ Press the **FUNCTION MODE** soft key |

| TURN | ▶ Function for machining mode: Press the **TURN** (Turning) or **MILL** (Milling) soft key |

If the machine tool builder has enabled kinematics selection, proceed as follows:

| SELECT | ▶ Press the **SELECT KINEMATICS** soft key |
| | ▶ Select the desired kinematics |

Example
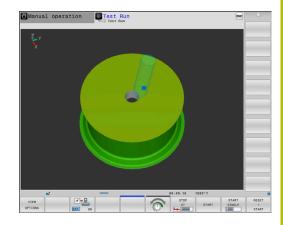
11 FUNCTION MODE TURN "AC_TABLE"	Activate turning mode
12 FUNCTION MODE TURN	Activate turning mode
13 FUNCTION MODE MILL "B_HEAD"	Activate milling mode

Graphic display of turning operations

You can simulate turning operations in **Test Run** mode. The requirement for this is a workpiece blank definition suitable for the turning process and option number 20.

> **i** The machining times determined using the graphic simulation do not correspond to the actual machining times. Reasons for this during combined milling-turning operations include the switching of operating modes.

9.3 Unbalance functions (option 50)

Unbalance while turning

General information

> ⚙ Refer to your machine manual!
>
> Unbalance functions are not required and available on all machine tool types.
>
> The unbalance functions described here are basic functions that are set up and adapted to the machine by the machine manufacturer. The scope and effect of the described functions may therefore vary from machine to machine. The machine manufacturer may also provide different unbalance functions.

In a turning operation, the tool is in a fixed position, whereas the rotary table and the clamped workpiece rotate. Depending on the size of the workpiece, the mass that is set in rotation can be very large. As the workpiece rotates, it creates an outward centrifugal force.

The centrifugal force that occurs basically depends on the rotational speed, the mass and the unbalance of the workpiece. When a body whose mass distribution is not rotationally symmetric is put into a rotary motion, this leads to an unbalance. If the mass object is rotating, this creates outward-acting centrifugal forces. If the rotating mass is evenly distributed, the centrifugal forces cancel each other out.

The unbalance is significantly influenced by the structural shape of the workpiece (e.g. an asymmetric pump housing) and by the fixtures. Because these conditions can often not be changed, compensate any existing unbalance by clamping a balancing weight. The controls provides the **MEASURE UNBALANCE** cycle for this purpose. The cycle determines the existing unbalance and calculates the mass and position of the required balancing mass.
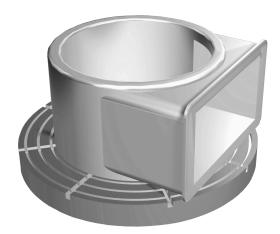
In the NC program, Cycle 892 **CHECK UNBALANCE** checks whether the entered parameters are exceeded.

> ### *NOTICE*
>
> **Caution: Significant property damage!**
>
> Very high physical forces are generated during turning, for example by high rotational speeds and heavy or unbalanced workpieces. Incorrect machining parameters, neglected unbalances or improper fixtures lead to an increased risk of accidents during machining!
>
> ▶ Clamp the workpiece in the spindle center
> ▶ Clamp workpiece securely
> ▶ Program low spindle speeds (increase as required)
> ▶ Limit the spindle speed (increase as required)
> ▶ Eliminate unbalance (calibrate)

ⓘ Operating notes:

- The rotation of the workpiece creates centrifugal forces that lead to vibration (resonance), depending on the unbalance. This vibration has a negative effect on the machining process and reduces the tool life.

- The removal of material during machining will change the mass distribution within the workpiece. This generates the unbalance, which is why an unbalance test is recommended even between the machining steps.

Unbalance Monitor function

The Unbalance Monitor function monitors the unbalance of a workpiece in turning mode. If a maximum unbalance limit specified by the machine manufacturer is exceeded, the control issues an error message and initiates an emergency stop. In addition, you can further decrease the permissible unbalance limit by setting the optional machine parameter **limitUnbalanceUsr** (no. 120101). If this limit is exceeded, the control issues an error message. Table rotation is not interrupted in this case. The control automatically activates the Unbalance Monitor function when you switch to turning mode. The unbalance monitor is effective until you switch back to milling mode.

📖 **Further information:** Cycle Programming User's Manual

Measure Unbalance cycle

ℹ	This cycle can only be run in turning mode. Activate **FUNCTION MODE TURN** beforehand.

To ensure maximum safety and minimum strain on the machine and workpiece during turning, you should check the unbalance of the clamped workpiece and compensate it with a balancing weight. The controls provides the **MEASURE UNBALANCE** cycle for this purpose.

The **MEASURE UNBALANCE** cycle determines the unbalance of the workpiece and calculates the mass and position of a balancing mass.

To determine the unbalance, proceed as follows:

▷ ► Shift the soft-key row in the Manual Operation mode

MANUAL CYCLES ► Press the **MANUAL CYCLES** soft key

► Press the **TURNING** soft key
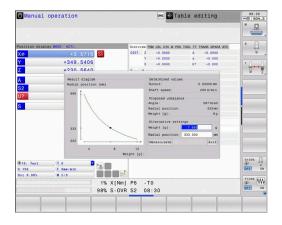
MEASURE UNBALANCE ► Press the **MEASURE UNBALANCE** soft key

► Enter the speed for unbalance detection

► Press NC start

> The cycle starts rotating the table at a low speed and gradually increases the speed up to the defined value.

> The control displays a window that shows the calculated mass and radial position of the balancing mass.

If you wish to use a different radial position or mass for the balancing mass, you can overwrite one value and have the other value recalculated automatically.

ℹ	Operating notes: ■ To compensate an unbalance, several balancing weights at different positions may be required. ■ After clamping a balancing weight, the unbalance must be checked again in a measurement.

Calibrate unbalance cycle

NOTICE

Danger of collision!

Changes to the calibration data can lead to undesired behavior. It is not recommended that the machine operator or NC programmer use the **CALIBRATE UNBALANCE** cycle. There is risk of collision during the execution of the function and during the subsequent processing!

▶ Use the function only in consultation with the machine tool builder

▶ Refer to the machine tool builder's documentation

The unbalance calibration is performed by the machine tool builder before shipping the machine. With unbalance calibration, the rotary table is operated at various speeds with a defined weight mounted at a defined radial position. The measurement is repeated with different weights.

9.4   Tools in turning mode (option 50)

Tool call

Just as in Milling mode, turning tools are called with the **TOOL CALL** function. You merely have to enter the tool number or tool name in the **TOOL CALL** block.

> 🛈   You can call and insert a turning tool both in Milling mode and in Turning mode.

Tool selection in the pop-up window

If you open a pop-up window for tool selection, the control marks all tools available in the tool magazine green.

In addition to the tool number and tool name, the control also shows the **ZL** and **XL** columns from the turning tool table.

Example

1 FUNCTION MODE TURN	Turning mode selection
2TOOL CALL "TRN_ROUGH"	Tool call
...	

Tool compensation in the NC program

With **FUNCTION TURNDATA CORR** you can define additional compensation values for the active tool. In the **TURNDATA CORR FUNCTION** you can enter delta values for tool lengths in the X direction **DXL** and in the Z direction **DZL**. The compensation values have an additive effect on the compensation values from the turning tool table.

With **FUNCTION TURNDATA CORR-TCS** you can define a cutter radius oversize **DRS**. This enables you to program an equidistant contour oversize. **DCW** allows you to compensate the recessing width of a recessing tool.

FUNCTION TURNDATA CORR is always effective for the active tool. A renewed **TOOL CALL** deactivates compensation again. When you exit the NC program (e.g. with PGM MGT), the control automatically resets the compensation values.

When using the **FUNCTION TURNDATA CORR** function, you can define via soft keys where the tool compensation will be effective:

- **FUNCTION TURNDATA CORR-TCS**: The tool compensation is effective in the tool coordinate system
- **FUNCTION TURNDATA CORR-WPL**: The tool compensation is effective in the workpiece coordinate system

> 🛈   Tool compensation **FUNCTION TURNDATA CORR-TCS** is always effective in the tool coordinate system, even during inclined machining.

> **i** During interpolation turning the functions **FUNCTION TURNDATA CORR** and **FUNCTION TURNDATA CORR-TCS** do not have any effect.
>
> If you want to compensate a turning tool during interpolation turning (Cycle 292), compensation needs to be performed in the cycle or in the tool table.
>
> **Further information:** Cycle Programming User's Manual

Defining the tool compensation

To define the tool compensation in the NC program, proceed as follows:

| SPEC FCT | ▶ Press the **SPEC FCT** key |

| TURNING PROGRAM FUNCTIONS | ▶ Press the **TURNING PROGRAM FUNCTIONS** soft key |

| FUNCTION TURNDATA | ▶ Press the **FUNCTION TURNDATA** soft key |

| TURNDATA CORR | ▶ Press the **TURNDATA CORR** soft key. |

> **i** As an alternative to tool compensation with **TURNDATA CORR**, you can use compensation tables.
>
> **Further information:** User's Manuals for Conversational Programming or ISO Programming

Example

| 21 FUNCTION TURNDATA CORR-TCS:Z/X DZL:0.1 DXL:0.05 |
| ... |

Tool data

You define turning-specific tool data in the turning tool table **TOOLTURN.TRN**.

The tool number saved in column **T** refers to the number of the turning tool in TOOL.T. Geometry values such as **L** and **R** from the TOOL.T are not effective with turning tools.

> ℹ The tool number in TOOLTURN.TRN must match the tool number of the turning tool in TOOL.T. If you enter or copy a new line you can then enter the corresponding number.

The tool length stored in the column **ZL** is saved by the control in the Q parameter Q114.

In addition you must identify turning tools in the tool table TOOL.T as turning tools. For this, in column TYP select the tool type **TURN** for the appropriate tool. If you require additional geometric data for a tool you can create further indexed tools for this.

You should give other tool tables that are to be archived or used for test runs different file names of your choice with the extension **.TRN**.

Proceed as follows to open the turning tool table:

> ▶ Select the machine operating mode, e.g. **Manual operation**

> ▶ Press the **TOOL TABLE** soft key
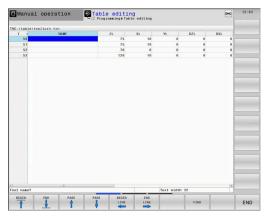
> ▶ Press the **TURNING TOOLS** soft key

> ▶ Edit the turning tool table: Set the **EDIT** soft key to **ON**

Tool data in the turning tool table

> **i** Below the table window the control displays dialog text, unit specification and input range for the specific input field.

> **i** In tool management, the control only displays the input fields related to the selected tool type.

Input parameters	Use	Input
T	Tool number: Must match the tool number of the turning tool in TOOL.T	-
NAME	Tool name: The control automatically takes on the tool name if you select the turning tool table in the tool table	**Input range: 32 characters max., only capital letters, no space characters**
ZL	Tool length 1 (Z direction)	**-99999.9999…+99999.9999**
XL	Tool length 2 (X direction)	**-99999.9999…+99999.9999**
YL	Tool length 3 (Y direction)	**-99999.9999…+99999.9999**
DZL	Delta value of tool length 1 (Z direction) is added to ZL	**-99999.9999…+99999.9999**
DXL	Delta value of tool length 2 (X direction) is added to XL	**-99999.9999…+99999.9999**
DYL	Delta value of tool length 3 (Y direction) is added to YL	**-99999.9999…+99999.9999**
RS	Tool tip radius: The control considers the tool tip radius in turning cycles and implements tool tip radius compensation when contours with radius compensation **RL** or **RR** were programmed	**-99999.9999…+99999.9999**
DRS	Delta value for tool tip radius: Cutter radius oversize is added to RS.	**-999.9999 to +999.9999**
TO	Tool orientation: Direction of tool tip	**1 to 9**
Angle of orientation	Spindle orientation angle: Angle of the indexable insert plane to the tool holder	**-360.0 to +360.0**
T-ANGLE	Setting angle for roughing and finishing tools	**0.0000 to +179.9999**
P-ANGLE	Point angle for roughing and finishing tools	**0.0000 to +179.9999**
CUTLENGTH	Cutting length of recessing tool	**0.0000 to +99999.9999**
CUTWIDTH	Width of the recessing tool	**0.0000 to +99999.9999**
DCW	Oversize f. recessing tool width	**-99999.9999…+99999.9999**
TYPE	Type of turning tool: Roughing tool **ROUGH**, finishing tool **FINISH**, thread tool **THREAD**, recessing tool **RECESS**, button tool **BUTTON**, groove turning tool **RECTURN**	**ROUGH**, **FINISH**, **THREAD**, **RECESS**, **BUTTON**, **RECTURN**
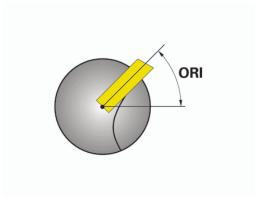
Orientation angle

With the spindle orientation angle **ORI** you define the angle position of the milling spindle for the turning tool.

> ℹ️ Operating notes:
> - The correct spindle angle is not only crucial for machining, but also for tool measurement.
> - The correct angle of orientation and the desired tool orientation of every newly defined tool should be checked.

Calculate the tool compensation

The measured compensation values **DXL** and **DZL** of a turning tool can be manually compensated in the tool management (option 93). The control automatically converts the input data into the tool coordinate system.

> ⚙️ Refer to your machine manual!
>
> Tool management is a machine-dependent function, which may be partly or completely deactivated.
> The machine tool builder defines the exact range of functions.

Dialog parameters	Description	Input
Compens. value WPL-Z	Measured error of the workpiece in Z direction	**-99999.9999...+99999.9999**
Compensation ØWPL-X	Measured error of the workpiece in X direction (diameter)	**-99999.9999...+99999.9999**
Inclination angle ß	Inclination angle during machining	**0.0000 to +179.9999**
Reverse the tool	Definition of whether the turning tool was used in a rotated position in the tool spindle.	-
Current value of DZL	Current calculated value for the tool	-
Current value of DXL	Current calculated value for the tool	-
New value of DZL	New calculated value for the tool	-
New value of DXL	New calculated value for the tool	-

Procedure

Proceed as follows to modify the compensation values:

▶ Select any machine operating mode, e.g. **Manual operation**

▶ Press the **TOOL TABLE** soft key

▶ Press the **TOOL MANAGEMENT** soft key

▶ Press the **FORM FOR TOOL** soft key

▶ Set the **EDIT** soft key to **ON**

▶ Use the arrow keys to select the **DXL** or **DZL** input field

▶ Press the **CALCULATE TOOL COMPENSTN.** soft key

> The control opens a pop-up window.

▶ Enter the compensation values

▶ Press the **APPLY** soft key if required

> The control loads the compensation values. You can then enter further compensation values.

▶ Press the **OK** soft key

> The control closes the pop-up window and saves the new compensation values to the tool table.

ℹ The control can describe the **DXL** and **DZL** columns using the touch probe cycles.

Further information: Cycle Programming User's Manual

Example

Input:
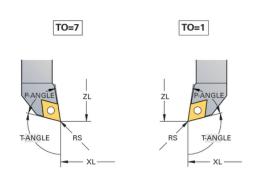
- **Compens. value WPL-Z**: 1
- **Compensation ØWPL-X**: 1
- **Inclination angle ß**: 90
- **Reverse the tool**: Yes

Result:

- **DZL**: +0.5
- **DXL**: +1
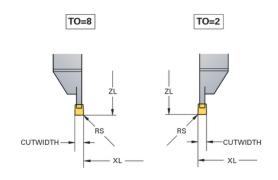
Tool data for turning tool

Input parameters	Use	Input
ZL	Tool length (#1)	Required
XL	Tool length (#2)	Required
YL	Tool length (#3)	Optional
DZL	Wear compensation **ZL**	Optional
DXL	Wear compensation **XL**	Optional
DYL	Wear compensation **YL**	Optional
RS	Cutting radius	Required
TO	Tool orientation	Required
Angle of orientation	Orientation angle	Required
T-ANGLE	Tool angle	Required
P-ANGLE	Point angle	Required
TYPE	Tool type	Required
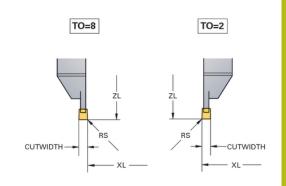
Tool data for recessing tools

Input parameters	Use	Input
ZL	Tool length (#1)	Required
XL	Tool length (#2)	Required
YL	Tool length (#3)	Optional
DZL	Wear compensation **ZL**	Optional
DXL	Wear compensation **XL**	Optional
DYL	Wear compensation **YL**	Optional
RS	Cutting radius	Required
TO	Tool orientation	Required
Angle of orientation	Orientation angle	Required
CUTWIDTH	Width of the recessing tool	Required
DCW	Oversize f. recessing tool width	Optional
TYPE	Tool type	Required
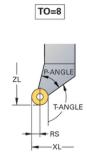
Tool data for groove turning tools

Input parameters	Use	Input
ZL	Tool length (#1)	Required
XL	Tool length (#2)	Required
YL	Tool length (#3)	Optional
DZL	Wear compensation **ZL**	Optional
DXL	Wear compensation **XL**	Optional
DYL	Wear compensation **YL**	Optional
RS	Cutting radius	Required
TO	Tool orientation	Required
Angle of orientation	Orientation angle	Required
CUTLENGTH	Cutting length of recessing tool	Required
CUTWIDTH	Width of the recessing tool	Required
DCW	Oversize f. recessing tool width	Optional
TYPE	Tool type	Required
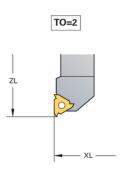
Tool data for button tools

Input parameters	Use	Input
ZL	Tool length (#1)	Required
XL	Tool length (#2)	Required
YL	Tool length (#3)	Optional
DZL	Wear compensation **ZL**	Optional
DXL	Wear compensation **XL**	Optional
DYL	Wear compensation **YL**	Optional
RS	Cutting radius	Required
TO	Tool orientation	Required
Angle of orientation	Orientation angle	Required
T-ANGLE	Tool angle	Required
P-ANGLE	Point angle	Required
TYPE	Tool type	Required
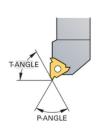
Tool data for threading tools

Input parameters	Use	Input
ZL	Tool length (#1)	Required
XL	Tool length (#2)	Required
YL	Tool length (#3)	Optional
DZL	Wear compensation **ZL**	Optional
DXL	Wear compensation **XL**	Optional
DYL	Wear compensation **YL**	Optional
TO	Tool orientation	Required
Angle of orientation	Orientation angle	Required
T-ANGLE	Tool angle	Required
P-ANGLE	Point angle	Required
TYPE	Tool type	Required

Tool radius compensation TRC

The tip of a lathe tool has a certain radius (**RS**). When machining tapers, chamfers and radii, this results in distortions on the contour because the programmed traverse paths refer to the theoretical tool tip S. TRC prevents the resulting deviations.

In the turning cycles the control automatically carries out tool radius compensation. In specific traversing blocks and within programmed contours, activate TRC with **RL** or **RR**.

The control checks the cutting geometry with the point angle **P-ANGLE** and the setting angle **T-ANGLE**. Contour elements in the cycle are processed by the control only as far as this is possible with the specific tool.

The control displays a warning when residual material is left behind due to the angle of the secondary cutting edges. You can suppress the warning with the machine parameter **suppressResMatlWar** (no. 201010).

<table>
<tr>
<td>🛈</td>
<td>

Programming notes:
- The direction of the radius compensation is not clear when the tool-tip position (**TO=2, 4, 6, 8**) is neutral. In this case, TRC is only possible within fixed machining cycles.

  The control can also run tool tip radius compensation during inclined processing.

  Active miscellaneous functions limit the possibilities here:
  - With **M128** tool-tip radius compensation is possible only in combination with machining cycles
  - **M144** or **FUNCTION TCPM** with **REFPNT TIP-CENTER** also allows tool-tip radius compensation with all positioning blocks, e.g. with **RL/RR**

</td>
</tr>
</table>

Theoretical tool tip

The theoretical tool tip is effective in the tool coordinate system. When the tool is inclined, the position of the tool tip rotates with the tool.

Virtual tool tip

Use **FUNCTION TCPM** with the selection **REFPNT TIP-CENTER** to activate the virtual tool tip. Correct tool data are the prerequisite for calculating the virtual tool tip.

The virtual tool tip is effective in the workpiece coordinate system. When the tool is inclined, the virtual tool tip remains unchanged as long as the tool orientation **TO** is the same. The control automatically switches the status display **TO** and thus also the virtual too tip if the tool leaves the angle range valid for **TO 1**, for example.

The virtual tool tip enables you to perform inclined paraxial longitudinal and transverse machining operations with high contour accuracy even without radius compensation.

10

Grinding

10.1 Grinding operations on milling machines (option 156)

Introduction

⚙ Refer to your machine manual!

Grinding must be configured and enabled by your machine tool builder. Some functions and cycles may not be available on your machine tool.

Special types of milling machines allow performing both milling and grinding operations. A workpiece can thus be machined completely on one machine, even if complex milling and grinding operations are required.

The term grinding encompasses many types of machining that differ in quite a few respects, e.g.:

- Jig grinding
- Cylindrical grinding
- Surface grinding

ℹ The TNC 640 currently features jig grinding.

Grinding tools

When managing grinding tools, other geometric descriptions than those for milling or drilling tools are required. The control provides special form-based tool management for grinding and dressing tools.

If grinding is enabled on your milling machine, (Option 156), the dressing function is also available. This means that you can shape or resharpen the grinding wheel in the machine.

Further information: "Entering tool data", Page 414

Jig grinding

> **ℹ** The control provides various cycles for the specific jig grinding or dressing movements.
>
> **Further information:** Cycle Programming User's Manual

Jig grinding means grinding of a 2-D contour. The tool movement in the plane may be superimposed by a reciprocation movement along the active tool axis.

On a milling machine, jig grinding will mainly be used for finishing a pre-machined contour with a grinding tool. There is not much of a difference between jig grinding and milling. Instead of a milling cutter, a grinding tool is used, such as a grinding pin or a grinding wheel. Jig grinding produces more precise results and a better surface quality than milling.

Machining is performed in milling mode, i.e. with **FUNCTION MODE MILL**.

Grinding cycles provide special movements for the grinding tool. A stroke or oscillating movement, the so-called reciprocating stroke, is superimposed with the movement in the working plane.

Grinding is also possible with a tilted working plane. The tool reciprocates along the active tool axis in the current working plane (WPL-CS).

Reciprocating stroke

For jig grinding, the movement of the tool in the plane can be superimposed with a stroke movement, the so-called reciprocating stroke. The superimposed stroke movement is effective in the active tool axis.

You define an upper and a lower stroke limit and can start and stop the reciprocating stroke and reset the corresponding values. The reciprocating stroke is effective until you stop it. **M30** will stop the reciprocating stroke automatically.

The control provides cycles for defining, starting, and stopping reciprocating strokes.

As long as the reciprocating stroke is active in a started NC program, you cannot change to the **Manual Operation** or **Positioning w/ Manual Data Input** operating modes.

> **ℹ** The reciprocating stroke continues to be effective during a programmed stop with **M0** and after the end of an NC block in the **Program run, single block** operating mode.

> **ℹ** The control does not support block scans while the reciprocating stroke is active.

Graphic display of the reciprocating stroke

The graphic simulation in the **Program run, single block** and **Program run, full sequence** operating modes shows the superimposed stroke movement.

Structure of the NC program

An NC program for grinding is structured as follows:

- Dressing of the grinding tool, if required
- Defining the reciprocating stroke
- If necessary, explicitly starting the reciprocating stroke
- Moving along the contour
- Stopping the reciprocating stroke

For the contour, you can select specific machining cycles, such as pocket, stud, or SL cycles.

With a grinding tool, the control behaves in the same way as with a milling cutter.

- If no cycle is programmed and you move along a contour whose smallest inside radius is smaller than the tool radius, the control will display an error message.
- If you machine with SL cycles, only those areas will be ground that are suitable for the given tool radius. In this case, the resulting contour will not be completely finished and may need to be reworked.

Further information: Cycle Programming User's Manual

Compensation in the grinding process

In order to achieve the required precision, you can use compensation tables during jig grinding.

Further information: User's Manual Conversational Programming

10.2 Tools for grinding (Option 156)

Grinding tool

In tool management, a special **grinding tool** tool type is available. You can call a grinding tool with **TOOL CALL**.

Example

```
5 TOOL CALL "GRIND" Z S15000 F200
```

The **DL** and **DR** compensation values are not compatible with grinding tools. If you program **DL** or **DR**, the control will display an error message.

For tool compensation, the control will automatically apply the compensation values stored in tool management. If you need to correct the grinding process, make sure to use the compensation tables.

Further information: User's Manuals for Conversational Programming or ISO Programming

Cutting edges of a grinding tool

Each grinding tool has several cutting edges, each with its own orientation and edge radius compensation. Select the desired cutting edge with Cycle 1030 **ACTIVATE WHEEL EDGE**.

For jig grinding, orientation 9 is mainly used. This orientation corresponds to the cutting edge of a milling cutter. For dressing, other wheel edges will be required as well. If you do not select a cutting edge, the control will automatically select orientation 9 for the grinding tool.

Further information: Cycle Programming User's Manual

Dressing tool

In tool management, a special **dressing tool** tool type is available. Define the dressing tool in tool management and call it with **TOOL CALL**.

Depending on the dressing tool, you need to specify a rotational speed when calling the tool.

> ⚙ Refer to your machine manual!
>
> The dressing tool will not be mounted to the spindle. You need to mount the dressing tool manually to a pocket defined by the machine tool builder.

If you require additional geometric data for a dressing tool, you can create further indexed tools for it.

Entering tool data

Define the specific data of grinding and dressing tools in the form-based tool management.

The control stores the tool data automatically in the **TOOLGRIND.GRD** file (for grinding tools) and in the **TOOLDRESS.DRS** file (for dressing tools).

To open tool management, proceed as follows:
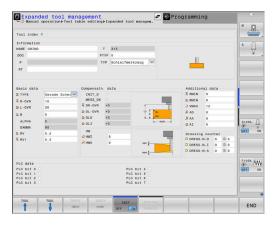
🖐	▶ Select the machine operating mode, e.g. **Manual operation**
TOOL TABLE	▶ Press the **TOOL TABLE** soft key
TOOL MANAGEMENT	▶ Press the **TOOL MANAGEMENT** soft key

You must indicate the tool type of grinding and dressing tools.

Proceed as follows:

FORM FOR TOOL	▶ Press the **FORM FOR TOOL** soft key
EDIT OFF [ON]	▶ Press the **EDIT** soft key
	▶ Enter the tool type.
	> The control displays the form for the specific tool data.

General tool data
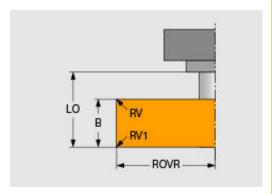
Input parameters	Use	Input
T	Number by which the tool is called in the NC program (e.g. 5, indexed: 5.2)	-
NAME	Name by which the tool is called in the NC program	**Input range: 32 characters max., only capital letters, no space characters**
DOC	Comment on the tool	**32 characters**
PTYP	Tool type for evaluation in the pocket table Refer to your machine manual! Function is defined by the machine tool builder!	**0...99**
P	Pocket number of the tool in the tool magazine	-
RT	Number of a replacement tool (**RT** for **R**eplacement-**T**ool) An empty field or input **0** means there is no replacement tool	**0...32767**
PLC data	Information on this tool that is to be sent to the PLC	**PLC bit 0..8**
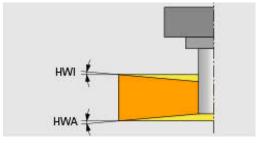
Tool data for grinding tools
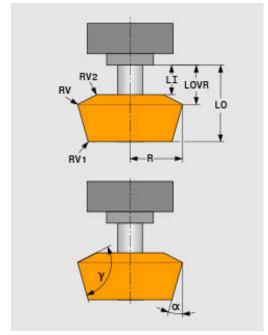
> ℹ️ In tool management, the control only displays the input fields related to the selected tool type.

Input parameters	Use
TYPE	Type of grinding tool
	Schleifstift, GRIND_M
	Schleifstift spezial, GRIND_MS
	Gerade Scheibe, GRIND_S
	Schräge Scheibe, GRIND_A
	Planscheibe, GRIND_P
R-OVR	Radius
L-OVR	Overhang
LO	Overall length
LI	Length to the inner edge
B	Width
G	Depth
R_SHAFT	Radius of the tool shank
ALPHA	Angle for the slant
GAMMA	Angle for the corner
RV	Radius at the edge of **L-OVR**
RV1	Radius at the edge of **LO**
RV2	Radius at the edge of **LI**
INIT_D	Initial dressing performed
MESS_OK	Grinding tool measured
DR_OVR	Radius compensation value
DL_OVR	Compensation value for the overhang
DLO	Compensation value for the total length
DLI	Compensation value for the length to the inner edge
HW	Relief cut exists
HWI	Angle for the relief cut on the inner side
HWA	Angle for the relief cut on the outer side
RMIN	Min. permissible radius
BMIN	Min. permissible width
VMAX	Maximum permissible cutting speed

Additional data for dressing

For grinding tools to be dressed, you need to define the following additional tool data:

Input parameters	Use
AD	Retraction amount at the diameter
AA	Retraction amount at the outer edge
AI	Retraction amount at the inner edge

The dressing cycles automatically use these retraction amounts There is no need to define approach or departing movements for the dressing contour.
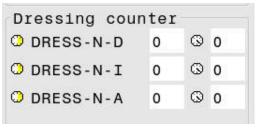
Input parameters	Use
DRESS-N-D	Dressing counter
	Diameter dressing
DRESS-N-I	Dressing counter
	Inner edge dressing
DRESS-N-A	Dressing counter
	Outer edge dressing

The left-hand side displays the nominal values programmed in the dressing cycle. The nominal value defines how often a dressing cycle will be called without being executed by the control.

The right-hand side displays the actual values, i.e. how often the dressing cycle has been skipped. You can change the actual value manually.
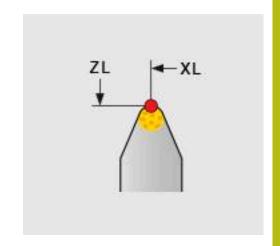
If the actual value and the nominal value are identical, the control will run the dressing cycle and delete the actual value.

Tool data for dressing tools

Input parameters	Use
ZL	Tool length 1 (Z direction)
XL	Tool length 2 (X direction)
YL	Tool length 3 (Y direction)
RS	Cutting radius
TYPE	Type of dressing tool
	Profile dresser (dressing diamond) DRESS_D
	Horn-type dresser DRESS_D2
	Dressing spindle DRESS_S
	Dress plate DRESS_P
	Dressing roller DRESS_R
TO	Tool orientation
DZL	Oversize for tool length 1 (Z direction)
DXL	Oversize for tool length 2 (X direction)
DYL	Oversize for tool length 3 (Y direction)
DRS	Cutting radius oversize

Grinding tool setup

HEIDENHAIN recommends the following sequence for grinding tool setup. It ensures that the control acquires the correct tool data to maintain the required grinding precision.

> ⚙   Depending on the tool to be used, you do not need to perform each step.

NOTICE

Danger of collision!

Changing or deleting grinding tool data directly in the tool table can result in undefined states at the machine. Incomplete tool definitions may lead to collisions between the tool and machine components or the workpiece!

▶ Make sure to adhere to the sequence when setting up grinding tools

▶ Only create grinding tools in tool management

▶ You can adjust the tool data using compensation in the cycles

---

To set up a grinding tool, proceed as follows:

▶ Enter the basic data for the grinding wheel in tool management
Further information: "Defining the basic data of the grinding wheel", Page 419

▶ Alternatively: Create an entry in the tool table with Cycle 1032 **GRINDING WHL LENGTH COMPENSATION** and Cycle 1033 **GRINDING WHL RADIUS COMPENSATION**

▶ Initial dressing
Further information: "Initial dressing", Page 419

> The control enables the checkbox for **INIT_D** in tool management.

▶ Measure the grinding wheel
Further information: "Measuring the grinding wheel", Page 420

> ℹ   Grinding tools that do not need dressing, such as diamond grinding wheels, can be measured directly. The control applies the compensations to the basic data.
>
> The **INIT_D** and **MESS_OK** flags are irrelevant in this case.

Defining the basic data of the grinding wheel
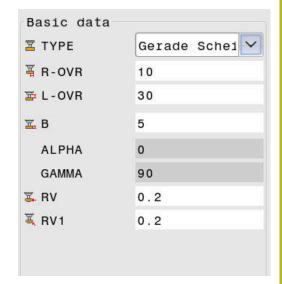
If the grinding wheel has not been created on the machine yet, the control requires its basic data first. You can either enter this data manually in tool management or have the grinding wheel measured in the machine.

If you enter the basic data manually, make sure not to enable the **INIT_D** checkbox. If the **INIT_D** flag is set, you cannot edit the basic data. While you can remove the checkmark manually, it is not possible to set it manually.

> **i** If you use Cycle 1032 **GRINDING WHL LENGTH COMPENSATION** to enter the basic data, the cycle will automatically delete any existing compensation values and set the required tool data.
>
> **Further information:** Cycle Programming User's Manual

NOTICE

Danger of collision!

For dressing, the basic data defines the distance between the dressing tool and the grinding tool. If the measured values are not accurate, especially if they are too small, a collision might occur during the first dressing run because the actual dressing value is larger than the programmed one.

▶ When manually inputting the basic values, enter values that are slightly larger than the measured ones

▶ Alternatively, have the grinding tool measured in the machine. The values are automatically entered in tool management

Initial dressing

Dressing a grinding tool for the first time is referred to as initial dressing. If a tool has not been dressed yet, the **INIT_D** flag is not set in tool management.

The relation between the grinding wheel and the dressing tool is not accurate before initial dressing. If you defined the grinding wheel larger than it is, the first dressing strokes will be performed in the air.

The control only considers the basic data of the grinding wheel for dressing.

Continue dressing until the programmed dressing amounts have actually been removed. Only then, the basic data of the grinding wheel match the dressing system used.

Further information: Cycle Programming User's Manual

Further information: User's Manuals for Conversational Programming or ISO Programming
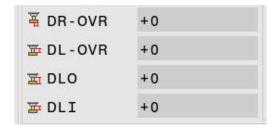
After dressing, the basic data you entered no longer match the actual measured values at the grinding tool. Correct this difference by measuring the grinding tool again after the dressing procedure.

Measuring the grinding wheel

If you performed initial dressing, the actual grinding tool dimensions probably no longer match the basic data. For this reason, have the grinding tool measured again.

To enter the correct compensation values into tool management, use Cycles 1032 **GRINDING WHL LENGTH COMPENSATION** and 1033 **GRINDING WHL RADIUS COMPENSATION**. After initial dressing, the control will only change the compensation values in order to avoid altering the relations to be used for further dressing procedures.

DR-OVR	+0
DL-OVR	+0
DLO	+0
DLI	+0

> ⚙ Refer to your machine manual!
>
> The control can set the **MESS_OK** flag, depending on how you measured the grinding tool. This flag indicates that the tool has been measured after initial dressing.

Further information: Cycle Programming User's Manual

11

MOD functions

11.1   MOD function
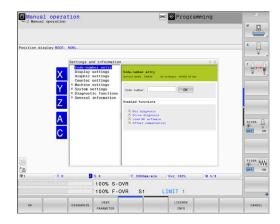
The MOD functions provide additional input possibilities and displays. In addition, you can enter code numbers to enable access to protected areas.

Selecting MOD functions

Open the pop-up window with the MOD functions:

MOD
▶ Press the **MOD** key
> The control opens a pop-up window displaying the available MOD functions.

Changing the settings

As well as with the mouse, navigation with the alphabetic keyboard is also possible in the MOD functions:
▶ Switch from the input area in the right window to the MOD function selections in the left window with the tab key
▶ Select MOD function
▶ Switch to the input field with the tab key or ENT key
▶ Enter value according to function and confirm with **OK** or make selection and confirm with **Apply**

> If there are multiple possible settings available, then you can show the selection box by pressing the **GOTO** key. Select the desired setting with the **ENT** key. If you do not wish to change the setting, close the window with the **END** key.

Exiting MOD functions

▶ Exit the MOD functions: Press the **END** soft key or the **END** key

Overview of MOD functions

The following functions are available independently of the selected operating mode:

Code-number entry

■ Code number

Display settings

■ Position displays

■ Unit of measure (mm/inch) for position display

■ Program input for MDI

■ Show time of day

■ Show the info line

Graphic settings

■ Model type

■ Model quality

Counter settings

■ Momentary count

■ Target value for counter

Machine settings

■ Kinematics

■ Traverse limits

■ Tool-usage file

■ External access

■ Set up wireless handwheel

■ Set up touch probes

System settings
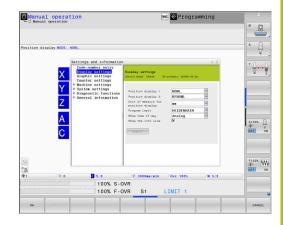
■ Set the system time

■ Define the network connection

■ Network: IP configuration

Diagnostic functions

■ Bus diagnosis

■ Diagnosis of drives

■ HEROS information

General information

■ Version information

■ License information

■ Machine times

11.2 Display software numbers

Application

The following software numbers are displayed on the control's screen after the **Software version** MOD function has been selected:

- **Control model**: Designation of the control (managed by HEIDENHAIN)
- **NC SW**: Number of the NC software (managed by HEIDENHAIN)
- **NCK**: Number of the NC software (managed by HEIDENHAIN)
- **PLC**: Number or name of the PLC software (managed by your machine manufacturer)

Your machine manufacturer can add further software numbers, e.g. from a connected camera.

In the **FCL Information** MOD function, the control shows the following information:

- **Development level (FCL=Feature Content Level):**
  Development level of the software installed on the control
  Further information: "Feature Content Level (upgrade functions)", Page 35

11.3 Entering the code number

Application

The control requires a code number for the following functions:

Function	Code number
Select user parameters	123
Configuring an Ethernet card	NET123
Enabling special functions for Q parameter programming	555343

Functions provided to the machine tool builder by the code number dialog

The two soft keys **OFFSET ADJUST** and **UPDATE DATA** are shown in the MOD menu of the control.

The **OFFSET ADJUST** soft key enables automatic determination and subsequent saving of an offset voltage required for analog axes.

> ⚙ Refer to your machine manual!
> This function may only be used by trained personnel!

The **UPDATE DATA** soft key enables the machine tool builder to install software updates on the control.

NOTICE

Caution: Data may be lost!

Not following the instructions when installing updates can cause a loss of data.

▶ Always follow the instructions when installing software updates!

▶ Refer to your machine manual.

11.4   Loading the machine configuration

Application

> **NOTICE**
>
> **Caution: Data may be lost!**
>
> The **RESTORE** function irrevocably overwrites the current machine configuration with the backup files. The control does not perform an automatic backup before the **RESTORE** function. The files are thus permanently gone.
>
> ▶ Perform a backup of the current machine configuration prior to the **RESTORE** function
>
> ▶ Use the function only in consultation with the machine tool builder

Your machine tool builder can provide you a backup with a machine configuration. After entering the keyword **RESTORE**, you can load the backup on your machine or programming station. Proceed as follows to load the backup:

▶ Enter the keyword **RESTORE** in the MOD dialog

▶ Select the backup file in the control's file manager (e.g., BKUP-2013-12-12_.zip)

> The control opens the pop-up window for the backup.

▶ Press Emergency Stop

▶ Press the **OK** soft key to start the backup process

11.5 Selecting the position display
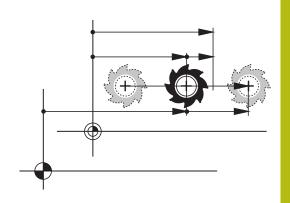
Application

You can influence the display of the coordinates for the **Manual operation** mode as well as for the **Program run, full sequence** and **Program run, single block** operating modes.

The figure on the right shows the different tool positions:

- Initial position
- Target position of the tool
- Workpiece datum
- Machine datum

You can select the following coordinates for the control's position displays:

Display	Function
NOML	Nominal position: The value currently commanded by the control
	ⓘ The NOML and ACTL displays differ solely with regard to following error.
ACTL	Actual position; current tool position
	⚙ Refer to your machine manual! Your machine tool builder defines whether the ACTL and NOML display deviates from the programmed position by the DL oversize of the tool call.
REF ACTL	Reference position; actual position relative to the machine datum
REF NOML	Reference position; nominal position relative to the machine datum
LAG	Servo lag; difference between nominal and actual positions
ACTDST	Distance remaining to the programmed position in the input coordinate system; difference between actual and target positions
	Examples with Cycle 11:
	▶ Scaling factor 0.2
	▶ L IX+10
	> The ACTDST display shows 10 mm.
	> The scaling factor does not have any influence.
	Example with Cycle 11 and tilled working plane:
	▶ Tilt A by 45°
	▶ Scaling factor 0.2
	▶ L IX+10
	> The ACTDST display shows 10 mm.
	> The scaling factor and the tilt do not have any influence.

Display	Function
REFDST	Distance remaining to the programmed position in the machine coordinate system; difference between actual and target positions
	Examples with Cycle 11
	▶ Scaling factor 0.2
	▶ L IX+10
	> The REFDST display shows 2 mm.
	> The scaling factor has an effect on the distance and thus on the display.
	Example with Cycle 11 and tilled working plane:
	▶ Tilt A by 45°
	▶ Scaling factor 0.2
	▶ L IX+10
	> The REFDST display shows 1.4 mm in the X and Z axes.
	> The scaling factor and the tilt have an effect on the distance and thus on the display.
M118	Traverse paths that were executed with handwheel superimpositioning function (**M118**)

> **ℹ** The **HR POS** tab of the additional status display (additional **VT** display) should be used for handwheel superimpositioning of the **Global Program Settings** function.

With the **Position display 1** MOD function, you can select the position display in the status display.

With the **Position display 2** MOD function, you can select the position display in the additional status display.

11.6 Setting the unit of measure

Application

With this MOD function, you can determine whether the control coordinates are displayed in millimeters or inches.

- Metric system: e.g. X = 15.789 (mm), the value is displayed to 3 decimal places
- Inch system: e.g. X = 0.6216 (inches), value is displayed to 4 decimal places

If you would like to activate the inch display, the control shows the feed rate in inch/min. In an inch program you must enter the feed rate larger by a factor of 10.

11.7 Graphic settings

With the **Graphic settings** MOD function, you can select the model type and model quality operating mode.

Proceed as follows to select **Graphic settings**:

▶ Select the **Graphic settings** group in the MOD menu
▶ Select the model type
▶ Select the model quality
▶ Press the **APPLY** soft key
▶ Press the **OK** soft key.

In the **Test Run** operating mode, the control displays icons of the active **Graphic settings**.

You have the following simulation parameters for the control's **Graphic settings**:

Model type

Icon	Choice	Properties	Application
	3-D	Very true to detail, heavy time and processor consumption	Milling with undercuts, milling-turning operations
	2.5 D	Fast	Milling without undercuts
	No model	Very fast	Line graphics

Model quality

Icon	Choice	Properties
	Very high	High data transfer rate, exact depiction of tool geometry, Block end points and block numbers can be displayed
	High	High data transfer rate, exact depiction of tool geometry
	Medium	Medium data transfer rate, approximation of tool geometry
	Low	Low data transfer rate, coarse approximation of tool geometry

11.8 Setting the counter

With the **Counter settings** MOD function, you can change the current count (actual value) and the target value (nominal value).

Proceed as follows to select **Counter settings**:

▶ Select the **Counter settings** group in the MOD menu
▶ Select the current count
▶ Select the target value for the counter
▶ Press the **APPLY** soft key
▶ Press the **OK** soft key

The control immediately takes over the selected value in the status display

Proceed as follows to change the **Counter settings** by soft key:

Soft key	Meaning
RESET	Reset count
+	Increase count
–	Lower count

You can also enter the values directly with a connected mouse.

Further information: "Defining a counter", Page 363

11.9   Changing the machine settings

Selecting the kinematics

> ⚙ Refer to your machine manual!
>
> Your machine tool builder configures and enables the **Kinematics selection** function.

NOTICE

Danger of collision!

All stored kinematics can also be selected as active machine kinematics. By this means, all manual movements and machining operations are executed using the selected kinematics. All subsequent axis movements pose a risk of collision!

▶ Use the **Kinematics selection** function only in the **Test Run** operating mode

▶ Use the **Kinematics selection** function for selecting the active machine kinematics only as needed

---

You can use this function to test NC programs whose kinematics does not match the active machine kinematics. If your machine tool builder saved different kinematic models in your machine and enabled them for selection, you can activate one of these kinematics configurations with the MOD function. When you select a kinematics model for the test run this does not affect machine kinematics.

> ℹ Ensure that you have selected the correct kinematics in the Test Run operating mode for checking your workpiece.

Entering traverse limits

🜨 Refer to your machine manual!

Your machine tool builder configures and enables the **Traverse limits** function.

The **Traverse limits** MOD function enables you to limit the actually usable traverse path within the maximum range of traverse. This enables you to define protection zones on each axis in order, for example, to protect an indexing head from collision.
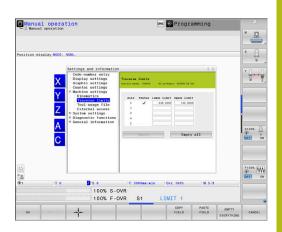
To enter traverse limits:

▶ In the MOD menu select the **Machine settings** group

▶ Select the **Traverse limits** menu

▶ Enter the values of the desired axes as a reference value or load the momentary position with the **actual position capture** soft key

▶ Press the **APPLY** soft key

> The control checks the entered values for validity.

▶ Press the soft key **OK**

ℹ Operating notes:

■ The protection zone becomes active automatically as soon as you have set a valid traverse limit in an axis. The settings are kept even after the control has been restarted.

■ You can only deactivate the protection zone by deleting all values or pressing the **EMPTY EVERYTHING** soft key.

Software limit switch with modulo axes:

If you set software limit switches for modulo axes, make sure to comply with the following basic conditions:

- The lower limit is greater than –360° and less than +360°
- The upper limit is not negative and less than +360°
- The lower limit is not greater than the upper limit
- The difference between the upper and the lower limit is less than 360°

If these conditions are not met, you will not be able to move the modulo axis. The TNC 640 displays an error message.

If modulo limit switches are active, a movement is allowed if the target position or an equivalent position is within the allowed range. Equivalent positions are positions that differ from the target positions by an offset of n · 360° (whereby n is any integer). The direction of motion is derived automatically because there is always only one equivalent position that the axis can move to—with the exception described below.

Example:

For the modulo axis C, the limit switches have been set to –80° and +80°. The axis is at 0°. If you program **L C+320** now, the C axis will move to –40°.

If an axis is positioned outside the limit switch range, it can only be moved towards the nearer limit switch.

Example:

The limit switches have been set to –90° and +90°. The C axis is at –100°.
In this case, the C axis must move in the positive direction with its next movement so that moving by **L C+I5** works, while **L C-I5** would result in a limit switch violation.

Exception:

The axis is positioned exactly in the middle of the forbidden range. Thus, the distance to both limit switches is the same. In this case, the movement can go to either direction. This results in the special situation that the axis can move to two equivalent positions if the target position is within the allowed range. In this case, the axis will be moved to the nearer equivalent position, i.e. the shorter path is used. If both equivalent positions are equidistant (i.e. 180° away), the control selects the direction of motion according to the programmed value.

Example:

The limit switches have been set to C–90°, C+90° and the axis is at 180°.
If you program **L C+0**, the C axis will move to 0. The same is true when programming **L C-360** etc. However, if you program **L C+360** (or L C+720, etc.) the C axis will move to 360°.

Generating a tool usage file

⚙️　　Refer to your machine manual!

　　　The tool usage test function must be enabled by your machine tool builder.

With the **Tool-usage file** MOD function, you can select whether the control never, once, or always creates a tool usage file.

Generate a tool usage file:

▶ In the MOD menu, select the **Machine settings** group

▶ Select the **Tool-usage file** menu

▶ Select the desired setting for the **Program Run, Full Sequence/ Single Block** and **Test Run** operating modes

▶ Press the **APPLY** soft key

▶ Press the **OK** soft key

Permitting or restricting external access

⚙️　　Refer to your machine manual!

　　　The machine tool builder can configure the external access options.

　　　Depending on the machine, you can grant or restrict access for an external diagnostics or commissioning software application using the **TNCOPT** soft key.

With the **External access** MOD function, you can grant or restrict access to the control. Once you have restricted external access, it is no longer possible to connect to the control and to exchange data over a network or over a serial connection (e.g., with the **TNCremo** software).
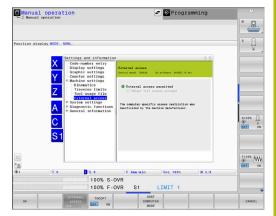
Proceed as follows to restrict external access:

▶ In the MOD menu, select the **Machine settings** group

▶ Select the **External access** menu

▶ Set the **EXTERNAL ACCESS ON/OFF** soft key to **OFF**

▶ Press the **OK** soft key

ℹ️　　Once the control is accessed externally, the following icon appears:

　　　⇄

Computer-specific access control

If your machine tool builder has set up computer-specific access control (machine parameter **CfgAccessControl** no. 123400), you can permit access for up to 32 connections authorized by you.

Proceed as follows:

▶ Select **Add** to create a new connection

> The control opens an input window for you to enter the connection data.

Access settings

Host name	Host name of the external computer
Host IP	Network address of the external computer
Description	Additional information (text is shown in the overview list)

Type:

Ethernet	Network connection
Com 1	Serial interface 1
COM 2	Serial interface 2

Access rights:

Inquire	For external access, the control opens a query dialog
Deny	Do not permit network access
Permit	Permit network access without query
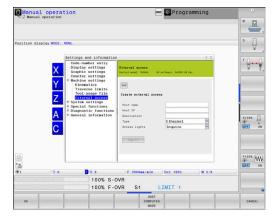
If you assign the **Inquire** access right to a connection, and if access is gained from this address, then the control opens a pop-up window. You must permit or deny external access in the pop-up window:

External access	Permission
Yes	Permit once
Always	Permit continuously
Never	Deny continuously
No	Deny once

i In the overview list, an active connection is shown with a green symbol.

Connections without access rights are shown gray in the overview list.

Host computer operation

⚙ Refer to your machine manual!
This function must be enabled and adapted by the machine tool builder.

With the **HOST COMPUTER MODE** soft key you transfer the command to an external host computer in order to transfer data to the control, for example.

The requirements to be met for starting host computer mode include the following:

- ▪ Dialogs, such as **GOTO** or **Block Scan** are closed
- ▪ No program run is active
- ▪ Handwheel is inactive

Proceed as follows to start the host computer mode:

▶ In the MOD menu, select the **Machine settings** group

▶ Select the **External access** menu

▶ Press the **HOST COMPUTER MODE** soft key

> The control shows an empty screen with the **Host computer is active** pop-up window.

⚙ Your machine tool builder can specify that the host computer mode can automatically be activated externally.

Exit the host computer mode as follows:

▶ Press the **HOST COMPUTER MODE** soft key again

11.10 Setting up touch probes

Introduction

The control allows you to set up and manage multiple touch probes. Depending on the type of touch probe, you have the following options for setting it up:

- TT tool touch probe with radio transmission: Setup via MOD dialog
- TT tool touch probe with cable or infrared transmission: Setup via MOD dialog or entry in the machine parameters
- TS 3-D touch probe with radio transmission: Setup via MOD dialog
- 3-D TS touch probe with cable or infrared transmission: Setup via MOD dialog, tool management, or touch probe table

    Further information: Cycle Programming User's Manual

Setting up a touch probe with radio transmission

> ⚙ Refer to your machine manual!
>
> In order for the control to recognize the touch probe with radio transmission, you will require an **SE 661** transceiver with EnDat interface.

Proceed as follows to open the setup dialog:

MOD	▶ Press the **MOD** key
	▶ Select the **Machine settings**
	▶ Select **Set up touch probes**
	> The control opens the device configuration on the third desktop.

On the left side, you will see the touch probes that have already been configured. If you are unable to see all of the columns, you can shift the view with the scroll bar or shift the dividing line between the left and right sides of the screen using the mouse.

Proceed as follows to set up a touch probe with radio transmission:

- ▶ Place the cursor on the row of the **SE 661**
- ▶ Select the radio channel

CONNECT NEW TCH PROBE	▶ Press the **CONNECT NEW TCH PROBE** soft key
	> The control displays the next steps in the dialog
	▶ Follow the instructions in the dialog:
	■ Remove the battery from the touch probe
	■ Insert the battery into the touch probe
	> The control connects to the touch probe and creates a new row in the table

Setting up a touch probe in the MOD dialog

You can set up a 3-D touch probe with cable or with infrared transmission either in the touch probe table, in tool management, or in the MOD dialog.

You can also define tool touch probes via the machine parameter **CfgTT** (No. 122700).

Proceed as follows to open the setup dialog:

MOD	▶ Press the **MOD** key
	▶ Select the **Machine settings**
	▶ Select **Set up touch probes**
	> The control opens the device configuration on the third desktop.

On the left side, you will see the touch probes that have already been configured. If you are unable to see all of the columns, you can shift the view with the scroll bar or shift the dividing line between the left and right sides of the screen using the mouse.

Setting up a 3-D touch probe

Proceed as follows to set up a 3-D touch probe:

CREATE TS ENTRY	▶ Press the **MAKE TT ENTRY** soft key
	> The control creates a new row in the table.
	▶ If necessary, highlight the row with the cursor
	▶ Enter the touch probe data on the right side
	> The control immediately saves the entered data in the touch probe table.

Setting up a tool touch probe

Proceed as follows to set up a tool touch probe

CREATE TT ENTRY	▶ Press the **MAKE TT ENTRY** soft key
	> The control opens a pop-up window.
	▶ Enter a unique name for the touch probe
	▶ Press **OK**
	> The control creates a new row in the table.
	▶ If necessary, highlight the row with the cursor
	▶ Enter the touch probe data on the right side
	> The control immediately saves the entered data in the machine parameters.

Touch probe with radio transmission configuration

The control displays the information on the individual touch probes on the right side of the screen. Some of this information is also visible and configurable for infrared touch probes.

Tab	TS 3-D Touch Probe	TT tool touch probe
Work data	Data from the touch probe table	Data from the machine parameters
Project information	Connection data and diagnostics functions	Connection data and diagnostics functions

You can change the data from the touch probe table by selecting the row with the cursor and overwriting the current value.

You can change the machine parameters only after first entering the code number.

Changing properties

Proceed as follows to change the touch probe properties:

▶ Place the cursor on the row for the touch probe

▶ Select the "Properties" tab

> The control shows the properties of the selected touch probe.

▶ Change the properties as desired per soft key

You have the following options depending on the row on which the cursor is located:

Soft key	Function
SELECT DEFLECTION	Select the probe signal
SELECT CHANNEL	Select the radio channel Select the channel with the best radio transmission and pay attention to overlaps with other machines or wireless handwheels.
CHANGE CHANNEL	Change the radio channel
REMOVE TCH. PROBE	Delete the touch probe data The control deletes the entry from the MOD dialog and the touch probe table or from the machine parameters.
EXCHANGE TCH. PROBE	Save a new touch probe in the current row The control automatically overwrites the serial number of the replaced touch probe with the new number.
SELECT SE	Select the SE transceiver
SELECT IR POWER	Select the strength of the infrared signal You only need to change the signal strength if there is interference.
SELECT RADIO POWER	Select the strength of the radio signal You only need to change the signal strength if there is interference.

The **Switching on/off** connection setting is preset based on the type of touch probe. Under **Deflection**, you can select how the touch probe is to transmit the signal when probing.

Deflection	Meaning
IR	Infrared probe signal
Radio	Radio probe signal
Radio + IR	The control selects the probe signal

You can activate the touch probe per soft key in the "Properties" tab (e.g., in order to test the radio connection)
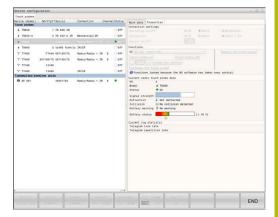
> **i** If you activate the touch probe's radio connection manually per soft key, then the signal remains unchanged even after a tool change. You must deactivate the radio connection manually again.

Current radio touch probe data
The control displays the following information in the "Current radio touch probe data" area:

Display	Meaning
NO.	Number in the touch probe table
Model	Type of touch probe
Status	Touch probe active or inactive
Signal strength	Display of the signal strength in the bar graphic
	The control shows the currently best-known connection as a complete bar
Deflection	Stylus deflected or not deflected
Collision	Collision or no collision recognized
Battery status	Display of the battery quality
	If the charge is less than the displayed bar, then the control outputs a warning.

11.11 Configuring the HR 550 FS radio handwheel

Application

ℹ️ This setup dialog is managed by the HEROS operating system.

After you have changed the conversational language on your control, you need to restart the control to enable the new language.

To configure the HR 550 FS radio handwheel, use the **SET UP WIRELESS HANDWHEEL** soft key. The following functions are available:

- Assigning the handwheel to a specific handwheel holder
- Setting the transmission channel
- Analyzing the frequency spectrum for determining the optimum transmission channel
- Selecting transmitter power
- Statistical information on the transmission quality

ℹ️ Any changes or modifications that were not expressly approved by the party responsible for conformity could void your authority to operate the device.

This device complies with Part 15 of the FCC Rules and the RSS standard(s) of Industry Canada for license-exempt equipment.

Operation is subject to the following conditions:

1 This device must not cause any harmful interference, and

2 this device must accept any interference received, including interference that may cause undesired operation.

Assigning the handwheel to a specific handwheel holder

▶ Make sure that the handwheel holder is connected to the control hardware.

▶ Place the wireless handwheel you want to assign to the handwheel holder in the handwheel holder

▶ Press the **MOD** key to select the MOD function

▶ Select the **Machine settings** menu

▶ Select the configuration menu for the radio handwheel: Press the **SET UP WIRELESS HANDWHEEL** soft key

▶ Click on the **Connect HW** button

> The control saves serial number of the inserted radio handwheel and shows it in the configuration window on the left next to the **Connect HW** button.

▶ Save the configuration and close the configuration menu: Press the **END** button

Setting the transmission channel

If the wireless handwheel is started automatically, then the control tries to select the transmission channel providing the best transmission signal. Proceed as follows if you want to set the radio channel yourself:

▶ Press the **MOD** key to select the MOD function

▶ Select the **Machine settings** menu

▶ Select the configuration menu for the radio handwheel: Press the **SET UP WIRELESS HANDWHEEL** soft key

▶ Click the **Frequency spectrum** tab

▶ Click the **Stop HW** button

> The control stops the connection to the wireless handwheel and determines the current frequency spectrum for all 16 available channels.

▶ Memorize the number of the channel with the least amount of radio traffic (smallest bar)

▶ Click the **Start handwheel** button to reactivate the radio handwheel

▶ Click the **Properties** tab

▶ Click the **Select channel** button

> The controls shows all available channel numbers

▶ Click the number of the channel that the control has found to have the least amount of radio traffic

▶ To save the configuration and exit the configuration menu, press the **END** button

Selecting the transmitter power

ℹ️ A reduction in transmission power decreases the range of the wireless handwheel.

▶ Press the **MOD** key to select the MOD function

▶ Select the **Machine settings** menu

▶ Select the configuration menu for the radio handwheel: Press the **SET UP WIRELESS HANDWHEEL** soft key

▶ Click the **Set power** button

> The control displays the three available power settings. Click on the desired setting.

▶ To save the configuration and exit the configuration menu, press the **END** button

Statistical data

To display the statistical data, proceed as follows:

▶ Press the **MOD** key to select the MOD function

▶ Select the **Machine settings** menu

▶ Select the configuration menu for the radio handwheel: Press the **SET UP WIRELESS HANDWHEEL** soft key

> The control displays the configuration menu with the statistical data.

Under **Statistics**, the control displays information about the transmission quality.

If the reception quality is poor so that a proper and safe stop of the axes cannot be ensured anymore, an emergency-stop reaction of the wireless handwheel is triggered.

The displayed value **Max. successive lost** indicates whether reception quality is poor. If the control repeatedly displays values greater than 2 during normal operation of the radio handwheel within the desired range of use, then there is a high risk of an undesired disconnection. This can be corrected by increasing the transmitter power or by changing to another channel with less radio traffic.

If this occurs, try to improve the transmission quality by selecting another channel or by increasing the transmitter power.

Further information: "Setting the transmission channel", Page 443

Further information: "Selecting the transmitter power", Page 443

11.12 Changing the system settings

Set the system time

With the **Set the system time** MOD function you can set the time zone, date and time manually or by means of an NTP server synchronization.

Proceed as follows to set the system time:

▶ In the MOD menu, select the **System settings** group

▶ Press the **SET DATE/ TIME** soft key

▶ In the **Time zone** area, select the desired time zone

▶ Press the **NTP on** soft key in order to select the **Set the time manually** entry

▶ Change the date and time as needed

▶ Press the **OK** soft key

To set the system time with the aid of an NTP server:

▶ In the MOD menu, select the **System settings** group

▶ Press the **SET DATE/ TIME** soft key

▶ In the **Time zone** area, select the desired time zone

▶ Press the **NTP off** soft key in order to select the **Synchronize the time over NTP server** entry

▶ Enter hostnames or the URL of an TNP server

▶ Press the **Add** soft key

▶ Press the **OK** soft key

11.13 Displaying operating times

Application
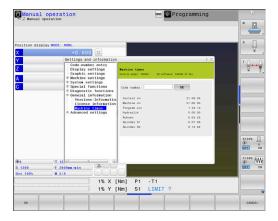
The **MACHINE TIME** MOD function enables you to see various types of operating times:

Operating time	Meaning
Control on	Operating time of the control since being put into service
Machine on	Operating time of the machine tool since being put into service
Program run	Duration of controlled operation since being put into service

⚙ Refer to your machine manual!

The machine tool builder can provide further operating time displays.

12

HEROS functions

12.1 Remote Desktop Manager (option 133)

Introduction

The Remote Desktop Manager enables you to display external computer units on the control screen that are connected via Ethernet, and to operate them via the control. You can also start programs specifically under HEROS or display web pages of an external server.

HEIDENHAIN offers you the IPC 6641 as a Windows computer. With the IPC 6641 Windows computer you can start and operate Windows-based applications directly from the control.

The following connection options are available:

■ **Windows Terminal Server (RemoteFX)**: Displays the desktop of a remote Windows computer on the control

■ **VNC**: Connection to an external computer. Displays the desktop of a remote Windows, Apple or Unix computer on the control

■ **Switch-off/restart of a computer**: Configure automatic shutdown of a Windows computer

■ **Web browser**: Available only to authorized specialists

■ **SSH**: Available only to authorized specialists

■ **XDMCP**: Available only to authorized specialists

■ **User-defined connection**: Available only to authorized specialists

> **i** HEIDENHAIN assures a functioning connection between HEROS 5 and the IPC 6641.
>
> No guarantee is given for other combinations and connections.

> **i** If you are using a TNC 640 with touch control, you can replace some keystrokes with gestures.
>
> **Further information:** "Operating the touchscreen", Page 533

Configuring connections – Windows Terminal Service (RemoteFX)

Configuring an external computer

ℹ️ You do not need additional software for your external computer to connect to the Windows Terminal Service.

Proceed as follows to configure the external computer, e.g. in the Windows 7 operating system:

▶ Press the Windows Start button and select **Control Panel** on the Start menu
▶ Select **System and Security**
▶ Select **System**
▶ Select **Remote settings**
▶ Under **Remote Assistance**, enable **Allow Remote Assistance connections to this computer**
▶ Under **Remote Desktop**, select **Allow connections from computers running any version of Remote Desktop**
▶ Press **OK** to confirm your settings

Configuring the control

Proceed as follows to configure the control:

▶ Press the **DIADUR** key to open the HEROS menu
▶ Select **Remote Desktop Manager**
> The control opens the **Remote Desktop Manager**.
▶ Press **New connection**
▶ Press **Windows Terminal Service (RemoteFX)**
> The control opens the **Select server operating system** pop-up window.
▶ Select the desired operating system
  ▪ Win XP
  ▪ Win 7
  ▪ Win 8.X
  ▪ Win 10
  ▪ Other Windows version
▶ Press **OK**
> The control opens the **Edit the connection** pop-up window.
▶ Edit the connection

Setting	Meaning	Input
Connection name	Name of the connection in the Remote Desktop Manager	Required
Restarting after end of connection	Behavior after disconnection: ■ Always restart ■ Never restart ■ Always after an error ■ Ask after an error	Required
Automatic starting upon login	Connection automatically established during control start-up	Required
Add to favorites	Connection icon in the task bar: ▶ Single click with the left mouse button > The control switches to the desktop of the connection. ▶ Single click with the right mouse button > The control displays the connection menu.	Required
Move to the following workspace	Number of the desktop for the connection, desktops 0 and 1 being reserved for the NC software Default setting: Third desktop	Required
Release USB mass memory	Permit access to connected USB mass memory devices	Required
Computer	Host name or IP address of the external computer HEIDENHAIN recommends the following setting for the IPC (6641): **IPC6641.machine.net** The host name **IPC6641** must be assigned to the IPC in the Windows operating system for this setting. ⓘ The **.machine.net** code is very important in this context. When **.machine.net** is specified, the control automatically searches the Ethernet interface **X116**, and not the interface **X26**—this reduces the time needed for access.	Required
User name	Name of the user	Required
Password	User password	Required
Windows domain	Domain of the external computer	Optional
Full-screen mode or **User-defined window size**	Size of the connection window	Required
Multimedia enhancements	Enables hardware acceleration when playing videos Certain formats, e.g. MP4 files, require the paid Fluendo Codec Pack ⚙ Any additional software will be installed by your machine tool builder.	Optional
Touchscreen input	Enables operation of multitouch systems and applications	Optional

Setting	Meaning	Input
Encryption	Sets the encryption that is appropriate for the selected Windows system	Required
	ⓘ If you activate the **Encryption** function, you need to remove the **-sec-tls -sec-nla** entries from the **Additional options** input field. If problems occur, try to set up a connection with the encryption function deactivated. The Windows log files are required for analysis.	
Color depth	Setting for the display of the external system on the control	Required
Locally effective keys	Shortcuts for automatically switching to the next active connection and workspace or desktop Default setting: ■ **Super_R** corresponds to the right DIADUR key and switches between the active connections ■ **F12** switches between the workspaces or desktops	Required
	ⓘ Touchscreens no longer include an **F12** key. Here, the vacant key between **PGM MGT** and **ERR** is used to switch the workspaces or desktops.	
	It is possible to adjust the default settings or to make additional entries	
Max. connection time (seconds)	Waiting time for connection If this timeout is exceeded, the connection could not be established	Required
Additional options	Available only to authorized specialists Additional command lines with transfer parameters	Required
	ⓘ If you activate the **Encryption** function, you need to remove the **-sec-tls -sec-nla** entries from the **Additional options** input field.	
Pass USB devices through	Passing the USB devices (e.g. 3-D mouse for operating CAD programs) connected to the control through to the Windows computer This requires installation of the Eltima EveUSB software on the Windows computer.	Optional
	ⓘ All USB devices that have been passed through are not available on the control while the connection to the Windows computer is active.	

For integrating the IPC 6641, HEIDENHAIN recommends the use of a RemoteFX connection.

When using a RemoteFX connection, the screen of the external computer is not mirrored, as with VNC. Instead, a separate desktop is opened. The desktop that is active on the external computer when establishing the connection is then locked and the user is logged off. This prevents two users from accessing the control simultaneously.

Configuring the connection – VNC

Configuring an external computer

> **i** You need an additional VNC server for your external computer when trying to connect to VNC.
>
> Install and configure the VNC server, e.g. the TightVNC server, before configuring the control.

Configuring the control

Proceed as follows to configure the control:

▶ Press the **DIADUR** key to open the HEROS menu
▶ Select **Remote Desktop Manager**
> The control opens the **Remote Desktop Manager**.
▶ Press **New connection**
▶ Press **VNC**
> The control opens the **Edit the connection** pop-up window.
▶ Edit the connection

Setting	Meaning	Input
Connection name:	Name of the connection in the Remote Desktop Manager	Required
Restarting after end of connection:	Behavior after disconnection: ▪ Always restart ▪ Never restart ▪ Always after an error ▪ Ask after an error	Required
Automatic starting upon login	Connection automatically established during control start-up	Required
Add to favorites	Connection icon in the task bar: ▶ Single click with the left mouse button > The control switches to the desktop of the connection. ▶ Single click with the right mouse button > The control displays the connection menu.	Required
Move to the following workspace	Number of the desktop for the connection, desktops 0 and 1 being reserved for the NC software Default setting: Third desktop	Required
Release USB mass memory	Permit access to connected USB mass memory devices	Required
Calculator	Host name or IP address of the external computer. In the recommended configuration of the IPC 6641, the IP address 192.168.254.3 is used.	Required
User name:	Name of the user to be logged on	Required
Password	Password for connecting to the VNC server	Required

Setting	Meaning	Input
Full-screen mode or **User-defined window size:**	Size of the connection window	Required
Permit further connections (share)	Additionally grant other VNC connections access to the VNC server	Required
View only	In display mode, the external computer cannot be operated	Required
Entries in the **Advanced options** area	Available only to authorized specialists	Optional

> ℹ️ If you are using **Extended Workspace Compact**, select the **Extended Workspace, Compact** function to enable the corresponding configuration for your connection.
>
> By selecting the **Extended Workspace, Compact** function, you automatically adapt the connections to the additional workspace.
>
> **Further information:** "Extended Workspace Compact", Page 63

With VNC, the screen of the external computer is mirrored directly. The active desktop at the external computer is not locked automatically.

With a VNC connection, it is also possible to completely shut down the external computer via the Windows menu. As the computer cannot be rebooted via a connection, it must actually be switched off and on again.

Shutting down or rebooting an external computer

NOTICE
Caution: Data may be lost!
If you do not shut down the external computer properly, data may be irreversibly damaged or deleted.
▶ Configure the automatic shutdown of the Windows computer

Proceed as follows to configure the control:

- ▶ Press the **DIADUR** key to open the HEROS menu
- ▶ Select **Remote Desktop Manager**
- ⏵ The control opens the **Remote Desktop Manager**.
- ▶ Press **New connection**
- ▶ Press **Switch-off/restart of a computer**
- ⏵ The control opens the **Edit the connection** pop-up window.
- ▶ Edit the connection

Setting	Meaning	Input
Connection name:	Name of the connection in the Remote Desktop Manager	Required
Restarting after end of connection:	Not necessary with this connection	–

Setting	Meaning	Input
Automatic starting upon login	Not necessary with this connection	–
Add to favorites	Connection icon in the task bar: ▶ Single click with the left mouse button > The control switches to the desktop of the connection. ▶ Single click with the right mouse button > The control displays the connection menu.	Required
Move to the following workspace	Not active with this connection	–
Release USB mass memory	Not advisable with this connection	–
Calculator	Host name or IP address of the external computer. In the recommended configuration of the IPC 6641, the IP address 192.168.254.3 is used.	Required
User name	User name to be used for establishing the connection	Required
Password	Password for connecting to the VNC server	Required
Windows domain:	Domain of the target computer, if required	Optional
Max. waiting time (seconds):	A shutdown of the control causes the Windows computer to shut down as well. Before the control displays the message **Now you can switch off.**, it waits for **<Timeout>** seconds. While waiting, the control checks whether the Windows computer is still accessible (port 445). If the Windows computer is switched off before the **<Timeout>** seconds have expired, the control will wait no longer.	Required
Additional waiting time:	Waiting time after the Windows computer has stopped being accessible. Windows applications may delay the shutdown of the computer after port 445 has been closed.	Required
Force	Close all programs on the Windows computer, even if dialogs are still open. If Force is not selected, Windows waits up to 20 seconds. This delays the shutdown process or the Windows computer is switched off before Windows has shut down.	Required
Restart	Reboot the Windows computer	Required
Run during restart	Reboot the Windows computer when the control reboots. Effective only if the control is rebooted using the shutdown icon at the bottom right in the taskbar or if it is rebooted as a result of a change in the system settings (e.g. network settings).	Required
Run during switch-off	The Windows computer is switched off when the control shuts down (no reboot). This is the standard scenario. Even the **END** key will then no longer trigger a reboot.	Required
Entries in the **Advanced options** area	Available only to authorized specialists	Optional

Starting and stopping the connection

Once a connection has been configured, this connection is shown as an icon in the Remote Desktop Manager window. Click the connection icon with the right mouse key to open a menu in which the display can be started and stopped.

If the desktop of the external connection or the external computer is active, all inputs from the mouse and the alphabetic keyboard are transmitted there.

When the HEROS 5 operating system is shut down, the control automatically terminates all connections. Please note, however, that only the connection is terminated, whereas the external computer or the external system is not shut down automatically.

Further information: "Shutting down or rebooting an external computer", Page 453

Proceed as follows to switch between the third desktop and the control interface:

- Use the right DIADUR key on the alphabetic keyboard
- Use the taskbar
- Use the operating mode key

12.2 Additional tools for ITCs

The following additional tools allow you to apply various settings for the touch screens on connected ITCs.

ITCs are industrial PCs without their own memory media, and therefore they do not have their own operating system. This feature is what makes ITCs different from IPCs.

ITCs are frequently used with large machinery, e.g. as a clone of the actual control system.

> ⚙ Refer to your machine manual!
>
> The machine manufacturer defines and configures the display and function of the connected ITCs and IPCs.

Additional tool	Application
ITC Calibration	4-point calibration
ITC Gestures	Configuration of gesture control
ITC touchscreen configuration	Selection of touch sensitivity

> ℹ The additional tools for the ITCs are only provided by the control in the taskbar with connected ITCs.

ITC Calibration

Using the additional tool **ITC Calibration,** you align the position for the mouse cursor displayed with the actual movement position of your finger.

Calibration using the additional **ITC Calibration** tool is recommended in the following cases:

- After replacing the touchscreen
- When changing the touch screen position (parallel axis error due to amended viewing angle)

Calibration involves the following steps:

▶ Start the tool in control using the task bar
> The ITC opens the calibration screen with four touch points in the corners of the screen
▶ Touch the four touch points shown one after the other
> The ITC closes the calibration screen once calibration has been successfully completed

ITC Gestures

Using the additional **ITC Gestures** tool, the machine manufacturer configures the gesture control on the touch screen.

> ⚙ Refer to your machine manual!
>
> This function may only be used with the permission of your machine manufacturer.

ITC touchscreen configuration

Using the additional **ITC Touchscreen Configuration** tool, you can select the touch sensitivity of the touch screen.

The ITC gives you the following options:

- **Normal Sensitivity (Cfg 0)**
- **High Sensitivity (Cfg 1)**
- **Low Sensitivity (Cfg 2)**

Use the **Normal Sensitivity (Cfg 0)** setting as standard. If you find it difficult to operate the equipment while wearing gloves in this setting, select the **High Sensitivity (Cfg 1)** setting.

> **ⓘ** If the ITC touch screen is not splash-proof, select the **Low Sensitivity (Cfg 2)** setting. This stops the ITC interpreting drops of water as touches.

Configuration involves the following steps:

- ▶ Start the tool in control using the task bar
- > The ITC opens a pop-up window with three options
- ▶ Select Touch Sensitivity
- ▶ Press the **OK** button
- > The ITC closes the pop-up window

12.3 Window manager

> ⚙ Refer to your machine manual!
>
> The machine tool builder determines the scope of function and behavior of the window manager.

The control features the Xfce window manager. Xfce is a standard application for UNIX-based operating systems, and is used to manage graphical user interfaces. The following functions are possible with the window manager:

- Display a task bar for switching between various applications (user interfaces)
- Manage an additional desktop, on which special applications from your machine manufacturer can run
- Control the focus between NC software applications and those of the machine manufacturer
- You can change the size and position of pop-up windows. It is also possible to close, minimize and restore pop-up windows

> ℹ The control shows a star in the upper left of the screen if an application of the window manager or the window manager itself has caused an error. In this case, switch to the window manager and correct the problem. If required, refer to your machine manual.

Overview of taskbar

In the task bar you can choose different workspaces by mouse click.

The control provides the following workspaces:

■ Workspace 1: Active operating mode

■ Workspace 2: Active programming mode

■ Workspace 3: CAD-Viewer or applications of the machine tool builder (optionally available)

■ Workspace 4: Display and remote control of external computer units (option 133) or applications of the machine tool builder (optionally available)

In addition, you can also select other applications from the task bar which you have started in parallel to the control software, e.g. the **TNCguide**.

> **ℹ** You can randomly move all open applications to the right of the green HEIDENHAIN symbol between the workspaces by pressing and holding the left mouse button.

Click the green HEIDENHAIN symbol to open a menu in which you can get information, make settings or start applications.

The following functions are available:

■ **About HEROS**: Open information about the operating system of the control

■ **NC Control**: Start and stop the control software (for diagnostic purposes only)

■ **Web Browser**: Start the web browser

■ **Touchscreen Calibration**: For calibrating the touch screen (only for touch operation)
Further information: "Touchscreen calibration", Page 547

■ **Touchscreen Configuration**: Adjust the screen properties (only for touch operation)
Further information: "Touchscreen configuration", Page 547
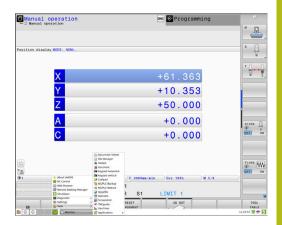
■ **Touchscreen Cleaning**: Lock the screen (only for touch operation)
Further information: "Touchscreen cleaning", Page 548

■ **Remote Desktop Manager** (option 133): Display and remote control of external computer units
Further information: "Remote Desktop Manager (option 133)", Page 448

- **Diagnostic**: Diagnostic applications
  - **GSmartControl**: Available only to authorized specialists
  - **HE Logging**: Define settings for internal diagnostic files
  - **HE Menu**: Available only to authorized specialists
  - **perf2**: Check processor load and process load
  - **Portscan**: Test active connections
    Further information: "Portscan", Page 462
  - **Portscan OEM**: Available only to authorized specialists
  - **RemoteService**: Start and stop remote maintenance
    Further information: "Remote Service", Page 463
  - **Terminal**: Enter and execute console commands
- **Settings**: Operating system settings
  - **Date/Time**: Set date and time
  - **Language/Keyboards**: Select system dialog language and keyboard version—the control overwrites the setting of the system dialog language when starting with the language setting of the machine parameter **CfgDisplayLanguage** (no. 101300)
  - **Network**: Define network settings
  - **Printer**: Configure and manage printer
    Further information: "Printer", Page 465
  - **Screensaver**: Setting the screensaver

    Further information: "Screensaver with lockout", Page 520
  - **Current User**: Show the current user

    Further information: "Current User", Page 524
  - **UserAdmin**: Configure user administration

    Further information: "Configuring the user administration", Page 496
  - **OEM Function Users**: Edit the OEM function users

    Further information: "HEIDENHAIN function users", Page 509
  - **SELinux**: Define safety software for Linux-based operating systems
  - **Shares**: Connect and manage external network drives
  - **State Reporting Interface** (Option 137): Activate the **SRI** and clear the status data
    Further information: "State Reporting Interface (option 137)", Page 467
  - **VNC**: Define the setting for external software accessing the control for e.g. maintenance work (**V**irtual **N**etwork **C**omputing)
    Further information: "VNC", Page 470
  - **WindowManagerConfig**: Available only to authorized specialists
  - **Firewall**: Configure the firewall
    Further information: "Firewall", Page 476
  - **HePacketManager**: Available only to authorized specialists
  - **HePacketManager Custom**: Available only to authorized specialists

- **Tools**: File applications
    - **Document Viewer**: Display and print files, e.g. PDF files
    - **File Manager**: Available only to authorized specialists
    - **Geeqie**: Open, manage, and print graphics
    - **Gnumeric**: Open, edit, and print tables
    - **Keypad**: Open virtual keyboard
    - **Leafpad**: Open and edit text files
    - **NC/PLC Backup**: Create backup file
      Further information: "Backup and restore", Page 473
    - **NC/PLC Restore**: Restore backup file
      Further information: "Backup and restore", Page 473
    - **QupZilla**: Alternative web browser for touch operation
    - **Ristretto**: Open graphics
    - **Screenshot**: Create screenshots
    - **TNCguide**: Call up help system
    - **Xarchiver**: Extract or compress directories
    - **Applications**: Supplementary applications
        - **Orage Calender**: Open calendar
        - **Real VNC viewer**: Define the setting for external software accessing the control for e.g. maintenance work (Virtual Network Computing)
    - **Shutdown**: Shut down the control

      Further information: "Switching or logging out users", Page 520

> **ⓘ** The applications available under tools can be started directly by selecting the corresponding file type in the file management of the control
> **Further information:** "Additional tools for management of external file types", Page 94

Portscan

The PortScan function allows the cyclic or manual searching for all open, incoming TCP and UDP listen ports on the system. All ports found are compared with whitelists. If the control finds a non-listed port it shows a corresponding pop-up window.

The HEROS **Diagnostic** menu contains the **Portscan** and **Portscan OEM** applications for this purpose. **Portscan OEM** is only executable after entering the machine manufacturer password.

The **Portscan** function searches for all open, incoming TCP and UDP ports on the system and compares them to four whitelists stored in the system:

- System-internal whitelists **/etc/sysconfig/portscan-whitelist.cfg** and **/mnt/sys/etc/sysconfig/portscan-whitelist.cfg**
- Whitelist for ports for machine manufacturer-specific functions, e.g. for Python applications, external applications: **/mnt/plc/etc/sysconfig/portscan-whitelist.cfg**
- Whitelist for ports with customer-specific functions: **/mnt/tnc/etc/sysconfig/portscan-whitelist.cfg**

For each entry, the whitelist specifies the type of port (TCP/UDP), the port number, the providing program, and optional comments. If the automatic port scan function is active, only ports listed in the whitelists can be open. Non-listed ports trigger a notification window.

The result of the scan is saved to a log file (LOG:/portscan/scanlog and LOG:/portscan/scanlogevil), and if new ports are found that are not listed in one of the whitelists these are displayed.

Manually starting Portscan

Proceed as follows to manually start the Portscan:

- ▶ Open the task bar at the bottom edge of the screen
  Further information: "Window manager", Page 458
- ▶ Press the green HEIDENHAIN button to open the **HEROS menu**
- ▶ Select the **Diagnostic** menu item
- ▶ Select the **Portscan** menu item
- ⊳ The control opens the **HEROS Portscan** pop-up window.
- ▶ Press the **Start** key

Cyclically starting Portscan

Proceed as follows to automatically start the Portscan cyclically:

- ▶ Open the task bar at the bottom edge of the screen
- ▶ Press the green HEIDENHAIN button to open the **HEROS menu**
- ▶ Select the **Diagnostic** menu item
- ▶ Select the **Portscan** menu item
- ⊳ The control opens the **HEROS Portscan** pop-up window.
- ▶ Press the **Automatic update on** key
- ▶ Set the time interval with the slider

Remote Service

In conjunction with the remote service setup tool, TeleService from HEIDENHAIN offers the possibility to establish encrypted end-to-end connections between a service computer and a machine.

To enable the HEIDENHAIN control to communicate with the HEIDENHAIN server, the control needs to be connected to the Internet.

Further information: "General network settings", Page 485

In its basic state the firewall of the control blocks all incoming and outgoing connections. Therefore, the firewall settings must be adjusted or the firewall must be deactivated for the duration of the service session.

Setting up the control

Proceed as follows to deactivate the firewall:

▶ Open the task bar at the bottom edge of the screen
▶ Press the green HEIDENHAIN button to open the **HEROS menu**
▶ Select the **Settings** menu item
▶ Select the **Firewall** menu item
> The control displays the **Firewall settings** dialog.
▶ Deactivate the firewall by removing the **Active** option in the **Firewall** tab.
▶ Press the **Apply** button to save the settings
▶ Press the **OK** button
> The firewall is disabled.

> **i** Remember to activate the firewall again at the end of the service session.

> **i** **Alternative procedure for deactivating the firewall**
>
> The remote diagnosis via the TeleService PC software uses the **LSV2** service and this service must therefore be permitted in the firewall settings.
>
> The following deviations from the firewall default settings are required:
>
> ▶ Set the method to **Permit some** for the **LSV2** service
> ▶ Enter the name of the service computer in the **Computer** column
>
> The access security is ensured through the network settings. The network security is the responsibility of the machine tool builder or the respective network administrator.
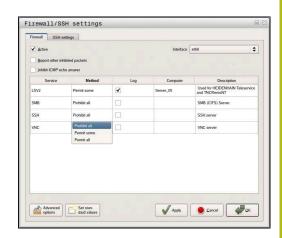
Automatic installation of a session certificate

With an NC software installation a temporary certificate is automatically installed on the control. An installation, also in the form of an update, may only be carried out by a service technician from the machine tool builder.

Manual installation of a session certificate

A new certificate must be installed if no valid session certificate is installed on the control. Contact the responsible service technician to determine which certificate you need. He will then provide you with a valid certificate file if necessary.

Proceed as follows to install the certificate on the control:
- ▶ Open the task bar at the bottom edge of the screen
- ▶ Press the green HEIDENHAIN button to open the **HEROS menu**
- ▶ Select the **Settings** menu item
- ▶ Select the **Network** menu item
- > The control displays the **Network settings** dialog.
- ▶ Select the **Internet** tab. The settings in the **Remote maintenance** field are configured by the machine tool builder.
- ▶ Press the **Add** button
- ▶ Select the file in the selection menu
- ▶ Press the **Open** button
- > The certificate is opened.
- ▶ Press the **OK** soft key.
- ▶ It may be necessary to restart the control to load the settings

Starting the service session

To start a service session, proceed as follows:
- ▶ Open the task bar at the bottom edge of the screen
- ▶ Press the green HEIDENHAIN button to open the **HEROS menu**
- ▶ Select the **Diagnostic** menu item
- ▶ Select the **RemoteService** menu item
- ▶ Enter the **Session key** of the machine tool builder

Printer

The **Printer** function in the HeROS menu enables you to add and manage printers.

Displaying the printer settings

Proceed as follows to access the printer settings:

▶ Open the task bar at the bottom edge of the screen
▶ Press the green HEIDENHAIN button to open the **HEROS menu**
▶ Select the **Settings** menu item
▶ Select the **Printer** menu item
> The control opens the **Heros Printer Manager** pop-up window.

The name of the printer is displayed in the input field.

Soft key	Meaning
CREATE	Creates the printer named in the input field
CHANGE	Modifies the properties of the selected printer
COPY	Creates the printer named in the input field with the attributes of the selected printer
	This can be useful if printing both portrait and landscape formats on the same printer
DELETE	Deletes the selected printer
UP	Selects the desired printer
DOWN	
STATUS	Displays status information of the selected printer
PRINT TEST PAGE	Prints a test page on the selected printer

For each printer, the following attributes can be set:

Setting	Meaning
Name of the printer	The printer name can be changed here.
Connection	Here, you can select the connection type ■ USB: The USB connection can be assigned here. The name is displayed automatically. ■ Network: Enter the network name or the IP address of the target printer here. In addition, specify the port of the network printer here (default: 9100) ■ Printer not connected
Timeout	Defines the delay to printing after the last change has been made to the file to be printed in PRINTER:. This can be useful if the file to be printed is populated with data by using FN functions, e.g. during probing.
Standard printer	Select to define the standard printer in case several printers are available. Is defined automatically when creating the first printer.
Settings for printing of text	These settings are applicable when printing text documents: ■ Paper size ■ Number of copies ■ Job name ■ Font size ■ Header ■ Print options (black and white, color, duplex)
Orientation	Portrait, landscape for all printable files
Expert options	Available only to authorized specialists

Print options:
■ Copying of the file to be printed in PRINTER:
  The file to be printed is automatically forwarded to the standard printer and deleted from the directory after the print job has been executed
■ Using the FN 16: F-PRINT function

List of printable files:
■ Text files
■ Graphic files
■ PDF files

> **i** The connected printer must be PostScript-enabled.

State Reporting Interface (option 137)

Introduction

With production lot sizes becoming ever smaller and products becoming increasingly customized, systems for production data acquisition are becoming increasingly important.

The operating resource data—as one of the most important parts of production data acquisition—describe the states of an operating resource along the time scale. For machine tools, the idle times and run times as well as information about pending faults are recorded. An evaluation per workpiece can also be made when the active NC programs are additionally taken into account.

One of the most common applications of production data acquisition is the determination of equipment effectiveness. The term Overall Equipment Effectiveness (OEE) is a measure of the value added of equipment. It provides information about the productivity of equipment as well as its losses at a glance.

With the **State Reporting Interface** (**SRI**), HEIDENHAIN offers a simple and reliable interface for acquiring the operating states of your machine.

In contrast to other interfaces, **SRI** also provides historical production data. Even if your company network is down for several hours, your valuable operating data will not be lost.

> **ℹ** A buffer memory that contains 2x 10 000 entries is available for storing the historical operating states. Each entry corresponds to a status change.

Configuring the control

Adjusting the firewall settings:

The **State Reporting Interface** uses **TCP Port 19090** to transfer the acquired operating states.

Accesses to the SRI from the company network (X26 connection) must be permitted in the firewall settings.

▶ Allow **SRI**

Further information: "Firewall", Page 476

> **ℹ** If the **SRI** is accessed locally via an IPC connected to the machine network (X116), eth0 (X26) can also remain locked.

Activating the **State Reporting Interface**:

In the control's factory default setting, the **SRI** is deactivated.

▶ Press the **DIADUR** key to open the HEROS menu

▶ Select the **Settings** menu item

▶ Select the **State Reporting Interface** menu item

▶ Activate the **State Reporting Interface** in the **SRI** pop-up window

Further information: "Overview of taskbar", Page 459

> **ℹ** To clear all previous operating states, use the **Clear historical data** button.

Acquiring operating states

The **State Reporting Interface** uses the **Hypertext Transfer Protocol** (**HTTP**) to transfer the operating states.

With the following **URL**s (**Uniform Resource Locator**), you can access the operating states of your control using any web browser.

- **http://<hostname>:19090/sri** for accessing all pieces of information (max. 20 000 entries)
- **http://<hostname>:19090/sri?lineno=<line>** for accessing the most recent pieces of information

Configuring **URL**:

▶ Replace the **<hostname>** by your control's network name

▶ Replace **<line>** by the first line to be retrieved

> The control transmits the requested data.

```
<html>
    <head></head>
    <body>
        <pre style="word-wrap: break-word; white-space: pre-wrap;">
            State Reporting Interface: 1.0.6
            HOST:     XXX
            HARDWARE: MC64XX 0.1
            SOFTWARE: 340590 10
            1 ; 2018-07-04 ; 09:52:22 ; TNC:\nc_prog\TS.h ; SUSPEND
            2 ; 2018-07-04 ; 09:52:28 ; TNC:\nc_prog\demo\Start_demo.h ; SUSPEND
            3 ; 2018-07-04 ; 09:52:30 ; TNC:\nc_prog\demo\Start_demo.h ; OPERATE
            4 ; 2018-07-04 ; 09:52:35 ; TNC:\nc_prog\demo\Start_demo.h ; ALARM
            5 ; 2018-07-04 ; 09:52:40 ; TNC:\nc_prog\demo\Start_demo.h ; SUSPEND
            6 ; 2018-07-04 ; 09:52:49 ; TNC:\nc_prog\$mdi.h ; SUSPEND
            7 ; 2018-07-04 ; 09:53:14 ; TNC:\nc_prog\demo\Start_demo.h ; SUSPEND
            8 ; 2018-07-04 ; 09:53:19 ; TNC:\nc_prog\demo\Start_demo.h ; OPERATE
            9 ; 2018-07-04 ; 09:53:24 ; TNC:\nc_prog\demo\Start_demo.h ; ALARM
        </pre>
    </body>
</html>
```

You will find the operating states in the `<body>` of the HTML file as **CSV** contents (**Comma Separated Values**).

CSV contents:

- Header

Designation	Meaning
State Reporting Interface:	Version of the interface.
	The version number should be taken into account in the evaluation of the data to ensure downward compatibility in your application.
SOFTWARE:	Software of the connected control.
HOST:	Full network name of the connected control.
HARDWARE:	Hardware of the connected control.

■ Operating data

Contents	Meaning
1	Consecutive number
2	
...	
2018-07-04	Date (yyyy-mm-dd)
09:52:22	Time (hh:mm:ss)
TNC:\nc_prog\TS.h	Selected or active NC program

States	Status:
■ **OPERATE**	■ Program run is active
■ **SUSPEND**	■ Program run stopped without error
■ **ALARM**	■ Program stopped due to an error

VNC

Use the **VNC** function to configure the behavior of the various VNC clients. This includes, for example, operation via soft keys, mouse and the alphabetic keyboard.

The control provides the following options:
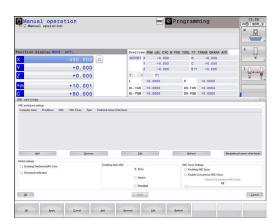
■ List of permitted clients (IP address or name)

■ Password for the connection

■ Additional server options

■ Additional settings for assigning the focus

> ⚙ Refer to your machine manual!
>
> For multiple clients or operating units, the focus assignment procedure depends on the design and the operating situation of the machine.
>
> This function must be adapted by your machine manufacturer.

Opening the VNC settings

Proceed as follows to open the VNC settings:

▶ Open the task bar at the bottom edge of the screen

▶ Press the green HEIDENHAIN button to open the **HEROS menu**

▶ Select the **Settings** menu item

▶ Select the **VNC** menu item

> The control opens the **VNC Settings** pop-up window.

The control provides the following options:

■ Add: Add new VNC viewer/client

■ Remove: Deletes the selected client. Only possible with manually entered clients.

■ Edit: Edit the configuration of the selected client

■ Update: Updates the display. Required with connection attempts during which the dialog is open.

VNC settings

Dialog	Option	Meaning
VNC participant settings	**Computer name:**	IP address or computer name
	VNC:	Connection of the client to the VNC viewer
	VNC Focus	The client participates in the focus assignment
	Model	■ Manual Manually entered client ■ Denied This client is not permitted to connect ■ Enable TeleService and IPC Client via TeleService connection ■ DHCP Other computer that obtains an IP address from this computer
Firewall warning		Warnings and information about if the VNC protocol has not been authorized for all VNC clients due to firewall settings on the control. **Further information:** "Firewall", Page 476.
Global settings	**Enable TeleService and IPC**	Connection is always permitted
	Password verification	The client must enter a password for verification. If this option is active the password must be entered when the connection is established.

Dialog	Option	Meaning
Enabling other VNC	**Deny**	Access generally denied to all other VNC clients.
	Inquire	During connection attempts a corresponding dialog is opened.
	Permitted	Access is generally granted to all other VNC clients.
VNC Focus Settings	**Enabling VNC focus**	Enable focus assignment for this system. Otherwise there is no central focus assignment. In the default setting, the focus is actively reassigned by the owner of the focus by clicking the focus symbol. This means that the owner of the focus must first release the focus by clicking the focus symbol before any other client can retrieve the focus.
	Enabling concurrency VNC focus	In the default setting, the focus is actively reassigned by the owner of the focus by clicking the focus symbol. This means that the owner of the focus must first release the focus by clicking the focus symbol before any other client can retrieve the focus. If concurrency focus is selected, any client can retrieve the focus at any time without having to wait for the current owner of the focus to release it.
	Timeout Concurrency VNC Focus	Time period within which the current owner of the focus can object to the focus being withdrawn or can prevent the reassignment of the focus. If a client requests the focus, a dialog in which the reassignment of focus can be refused appears on all clients' screens.
Focus symbol		Current status of VNC focus on the respective client: Focus is owned by other client. Mouse and alphabetic keyboard are disabled.
		Current status of VNC focus on respective client: Focus is owned by current client. Entries can be made.
		Current status of VNC focus on the respective client: Request by the owner of the focus to give the focus to another client. Mouse and alphabetic keyboard are disabled until the focus is assigned unambiguously.

If **Enable concurrency VNC focus** is selected, a pop-up window appears. This dialog makes it possible to refuse that the focus be given to the requesting client. If this does not occur, the focus changes to the requesting client after the set time limit.

Backup and restore

With the **NC/PLC Backup** and **NC/PLC Restore** functions you can back up and restore individual folders or the complete **TNC** drive. You can save the backup files locally, on a network drive, or to USB storage devices.

The backup program generates a ***. tncbck** file that can also be processed by the PC tool TNCbackup (part of TNCremo). The Restore program can restore these files as well as those from existing TNCbackup programs. If you select a *. tncbck file in the File Manager of the control, the **NC/PLC Restore** program will be launched automatically.

Backup and restore is subdivided into several steps. Navigate between these steps with the **FORWARD** and **BACK** soft keys. Specific actions for steps are selectively displayed as soft keys.

Open NC/PLC Backup or NC/PLC Restore

Proceed as follows to open the functions:

▶ Open the task bar at the bottom edge of the screen
▶ Press the green HEIDENHAIN button to open the **HEROS menu**
▶ Select the **Tools** menu item
▶ Open the **NC/PLC Backup** or **NC/PLC Restore** menu item
> The control opens the pop-up window.

Backing up data

⚙ **QR** parameters will be included in backups.

If the machine tool builder did not define a specific path, the control will save the **QR** parameter values in the following path: **SYS:\runtime\sys.cfg**. This partition will only be backed up in full backups.

Machine tool builders can use the following optional machine parameters to specify the paths:

- **pathNcQR** (no. 131201)
- **pathSimQR** (no. 131202)

If the machine tool builder used the optional machine parameters to specify a path on the TNC partition, you can perform a backup with the **NC/PLC Backup** functions without entering a code number.

To backup data from the control, proceed as follows:

▶ Select **NC/PLC Backup**
▶ Select
type
   - Back up the **TNC** partition
   - Back up the directory tree: Select the directory for backup in the file management
   - Backup machine configuration (for machine tool builders only)
   - Complete backup (for machine tool builders only)
   - Comment: Freely configurable comment for the backup
▶ Select the next step with the **FORWARD** soft key
▶ Stop the control if required with the **STOP NC SOFTWARE** soft key
▶ Define the exclusion rules
   - User preset rules
   - Write own rules to the table
▶ Select the next step with the **FORWARD** soft key
> The control generates a list of files for backing up.
▶ Check the list. Deselect files if necessary.
▶ Select the next step with the **FORWARD** soft key
▶ Enter the name of the backup file
▶ Select the storage path
▶ Select the next step with the **FORWARD** soft key
> The control generates the backup file.
▶ Confirm with the **OK** soft key
> The control concludes the backup process and restarts the NC software.

Restore data

> | **NOTICE** |
> | :---: |
> | **Caution: Data may be lost!** |
> | When you restore data (Restore function), any existing data will be overwritten without a confirmation prompt. Existing data is not automatically backed up by the control before running the restore process. Power failures or other problems can interfere with the data restore process. As a consequence, data may be irreversibly damaged or deleted. |
> | ▶ Before starting the data restore process, make a backup of the existing data |

To restore the data proceed as follows:

▶ Select **NC/PLC Restore**
▶ Select the archive to be restored
▶ Select the next step with the **FORWARD** soft key
> The control generates a list of files for restoring.
▶ Check the list. Deselect files if necessary.
▶ Select the next step with the **FORWARD** soft key
▶ Stop the control if required with the **STOP NC SOFTWARE** soft key
▶ Extract the archive
> The control restores the files.
▶ Confirm with the **OK** soft key
> The control restarts the NC software.

12.4 Firewall

Application

You can set up a firewall for the primary network interface of the control. It can be configured so that incoming network traffic is blocked and/or a message is displayed, depending on the sender and the service. The firewall cannot be started for the secondary network interface of the control.

Once the firewall has been activated, a symbol appears at the lower right in the taskbar. The symbol changes depending on the safety level that the firewall was activated with, and informs about the level of the safety settings:

Icon	Meaning
	Firewall protection does not yet exist although it has been activated according to the configuration. This can happen, for example, if PC names for which there are no equivalent IP addresses as yet were used in the configuration.
	Firewall active with medium security level
	Firewall active with high safety level. (all services except for SSH are blocked)

> **i** Have your network specialist check and, if necessary, change the standard settings.

Configuring the firewall

Make your firewall settings as follows:

▶ Use the mouse to open the taskbar at the bottom edge of the screen

▶ Press the green HEIDENHAIN button to open the HEROS menu

▶ Select the **Settings** menu item

▶ Select the **Firewall** menu item

HEIDENHAIN recommends activating the firewall with the prepared default settings:

▶ Select the **Active** option to enable the firewall

▶ Press the **Set standard values** button to activate the default settings recommended by HEIDENHAIN.

▶ Confirm the changes with **Apply**

▶ Close the dialog with **OK**.

Firewall settings

Option	Meaning
Active	Switching the firewall on and off
Interface	Selection of the **eth0** interface usually corresponds to X26 of the MC main computer. **eth1** corresponds to X116. You can check this in the network settings on the Interfaces tab. For main computer units with two Ethernet interfaces, the DHCP server for the machine network is active for the second (non-primary) interface by default. With this setting the firewall cannot be activated for **eth1** because the firewall and DHCP server mutually exclude each other.
	ℹ️ To configure the sandbox, you use the optional interface brsb0. **Further information:** "Sandbox tab", Page 491
Report other inhibited packets	Firewall active with high safety level. (all services except for SSH are blocked)
Inhibit ICMP echo answer	If this option is set, the control no longer responds to a PING request
Service	This column contains the short names of the services that are configured with this dialog. For the configuration it is not important here whether the services themselves have been started
	■ **DNC** designates the service the DNC server provides via the RPC protocol for external applications that were developed with the RemoTools SDK (port 19003)
	📖 For more detailed information, consult the RemoTools SDK manual.
	■ **LDAPS** includes the server on which the user data and the user administration configuration are saved.
	■ **LSV2** includes the functionality for **TNCremo**, Teleservice and other HEIDENHAIN PC Tools (port 19000)
	■ **OPCUA** designates the service provided by the HEIDENHAIN OPC UA NC Server (port 4840)
	■ **SMB** only refers to incoming SMB connections, i.e. if a Windows share is created on the NC. Outgoing SMB connections (i.e. if a Windows share is connected on the NC) cannot be prevented.
	■ **SRI** refers to the connections that are used in conjunction with the acquisition of operating states through the **State Reporting Interface** option.
	■ **SSH** stands for the Secure Shell protocol (port 22). As of HEROS 504, LSV2 can be executed securely via this SSH protocol while user administration is active. **Further information:** "User authentication from external applications", Page 514
	■ **VNC** protocol means access to the screen contents. If this service is blocked, the screen content can no longer be accessed, not even with the TeleService programs from HEIDENHAIN (e.g. screenshot). If this service is blocked, the VNC configuration dialog shows a warning from HEROS that VNC is disabled in the firewall.

Option	Meaning
Method	Use **Method** to configure whether the service is inaccessible for all (**Prohibit all**), accessible for all (**Permit all**) or only accessible for individuals (**Permit some**). If you set **Permit some** you must also specify the computer (under Computer) that you wish to grant access to the respective service. If you do not specify any computer under **Computer**, the setting **Prohibit all** will automatically become active when the configuration is saved.
Log	If **Log** is activated, a **red** message is output if a network packet for this service was blocked. A (blue) message is output if a network packet for this service was accepted
Calculator	If the setting **Permit some** is selected under **Method**, the relevant computers can be specified here. The computers can be entered with their IP addresses or host names separated by commas. If a host name is used, the system checks upon closing or saving of the dialog whether the host name can be translated into an IP address. If this is not the case, an error message is displayed and the dialog does not terminate. If a valid host name is specified, this host name is translated into an IP address each time the control is started. If a computer that was entered with its name changes its IP address, you may have to restart the control or formally change the firewall configuration to ensure that the control uses the new IP address for a host name in the firewall.
Advanced options	These settings are only intended for your network specialists
Set standard values	Resets the settings to the default values recommended by HEIDENHAIN
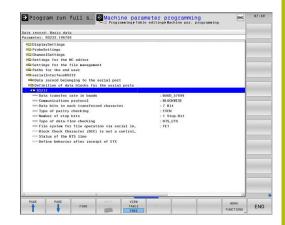
12.5 Setting up data interfaces

Serial interfaces on the TNC 640

The TNC 640 automatically uses the LSV2 transmission protocol for serial data transfer. The LSV2 protocol is prescribed and cannot be modified apart from setting the baud rate (machine parameter **baudRateLsv2,** no. 106606). You can also define another type of data transfer (interface). The settings described below are therefore effective only for the respective newly defined interface.

Application

To set up a data interface, press the **MOD** key. Enter the code number 123. In the **CfgSerialInterface** (no. 106700) machine parameter, you can enter the following settings:

Setting the RS-232 interface

Open the RS232 folder. The control then displays the following settings:

Set BAUD RATE
(baud rate no. 106701)

You can set the BAUD RATE (data transfer speed) from 110 to 115 200 baud.

Set protocol
(protocol no. 106702)

The data transfer protocol controls the data flow of a serial transmission (comparable to MP5030 of the iTNC 530).

> ℹ️ Operating notes:
> - The **BLOCKWISE** setting designates a type of data transfer in which the data is transferred grouped in blocks.
> - The **BLOCKWISE** setting does **not** correspond to the data reception in blocks nor to the simultaneous execution of older contouring controls in blocks. This function is no longer available for current controls.

Data transmission protocol	Selection
Standard data transmission (transmission line-by-line)	STANDARD
Packet-based data transfer	BLOCKWISE
Transmission without protocol (only character-by-character)	RAW_DATA

Set data bits
(dataBits no. 106703)

By setting the data bits you define whether a character is transmitted with 7 or 8 data bits.

Check parity
(parity no. 106704)

The parity bit helps the receiver to detect transmission errors. The parity bit can be formed in three different ways:

- No parity (NONE): There is no error detection
- Even parity (EVEN): Here there is an error if the receiver finds that it has received an odd number of set bits
- Odd parity (ODD): Here there is an error if the receiver finds that it has received an even number of set bits

Set stop bits
(stopBits no. 106705)

The start bit and one or two stop bits enable the receiver to synchronize each transmitted character during serial data transmission.

Set handshake
(flowControl no. 106706)

By handshaking, two devices control data transfer between them.
A distinction is made between software handshaking and hardware
handshaking.

- No data flow checking (NONE): Handshaking is not active
- Hardware handshaking (RTS_CTS): Transmission stop is active
  through RTS
- Software handshaking (XON_XOFF): Transmission stop is active
  through DC3 (XOFF)

File system for file operation
(fileSystem no. 106707)

In **fileSystem** you define the file system for the serial interface.
This machine parameter is not required if you don't need a special
file system.

- EXT: Minimum file system for printers or non-HEIDENHAIN
  transmission software. It corresponds to the EXT1 and EXT2
  operating modes on older HEIDENHAIN controls.
- FE1: Communication with the TNCserver PC software or an
  external floppy disk unit.

Block check character
(bccAvoidCtrlChar no. 106708)

With Block Check Character (optional) no control character, you
determine whether the checksum can correspond to a control
character.

- TRUE: The checksum does not correspond to a control character
- FALSE: The checksum can correspond to a control character

Condition of RTS line
(rtsLow no. 106709)

With the state of the RTS line (optional), you can define whether
the **LOW** level is active in idle state.

- TRUE: Level is **LOW** in idle state
- FALSE: Level is not **LOW** in idle state

Define behavior after receipt of ETX
(noEotAfterEtx no. 106710)

With define behavior after reception of ETX (optional) you
determine whether the EOT character is sent after the ETX
character was received.

- TRUE: The EOT character is not sent
- FALSE: The EOT character is sent

Settings for the transmission of data using PC software TNCserver

Apply the following settings in machine parameter **RS232** (no. 106700):

Parameters	Selection
Data transfer rate in baud	Has to match the setting in TNCserver
Data transmission protocol	BLOCKWISE
Data bits in each transferred character	7 bits
Type of parity checking	EVEN
Number of stop bits	1 stop bit
Specify type of handshake:	RTS_CTS
File system for file operations	FE1

Setting the operating mode of the external device (fileSystem)

> 🛈 The **load all programs**, **load offered program**, and **load directory** functions are not available in the **FE2** and **FEX** operating modes.

Icon	External device	Operating mode
🖥	PC with **TNCremo** software	LSV2
💾	HEIDENHAIN floppy disk units	FE1
🖧	Non-HEIDENHAIN devices such as printers, scanners, punchers, PC without **TNCremo**	FEX

Software for data transfer

For data transfer to or from the control, you should use the **TNCremo** software. With **TNCremo**, data transfer is possible with all HEIDENHAIN controls via the serial interface or the Ethernet interface.

> **ⓘ** You can download the current version of the **TNCremo** software from the HEIDENHAIN homepage.

System requirements for TNCremo:
- Operating system
  - Windows 7
  - Windows 8
  - Windows 8.1
  - Windows 10
- 2 GB RAM
- 5 MB of free disk space
- An available serial interface or connection to the TCP/IP network

Installation under Windows
▶ Open the file manager (Explorer) and start the SETUP.EXE installer
▶ Follow the setup program instructions

Starting TNCremo under Windows
▶ Click <Start>, <All Programs>, <HEIDENHAIN>, <**TNCremo**>
▶ Alternatively, double-click the TNCremo icon on your desktop

Data transfer between the control and TNCremo

Check whether the control is connected to the correct serial port on your PC or to the network.

Once you have started the **TNCremo** software, you will see a list of all files that are stored in the active directory in the upper section of the main window **1**. Using <File>, <Change directory>, you can select any drive or another directory on your PC.

If you want to control data transfer from the PC, establish the connection with your PC in the following manner:

▶ Select <File>, <Setup connection>. **TNCremo** now receives the file and directory structure from the control and displays this in the lower part of the main window **2**

▶ To transfer a file from the control to the PC, select the file in the control window per mouse click and move the highlighted file into the PC window while holding down the mouse button **1**

▶ To transfer a file from the PC to the control, select the file in the PC window per mouse click and move the highlighted file into the control window while holding down the mouse button **2**

If you want to control data transfer from the control, establish the connection with your PC in the following manner:

▶ Select <Extras>, <TNCserver>. **TNCremo** then starts in server mode and can receive data from the control or send data to the control

▶ You can now call the file management functions on the control by pressing the **PGM MGT** key in order to transfer the desired files
   Further information: "Data transfer to or from an external data carrier", Page 90

🛈 If you have exported a tool table from the control, then the tool types are converted to tool type numbers.
   Further information: "Available tool types", Page 157
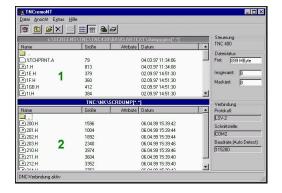
End TNCremo

Select <File>, <Exit>

🛈 You can open the context-sensitive help function of the **TNCremo** software by pressing the **F1** key.

12.6 Ethernet interface

Introduction

The control is provided with an Ethernet card as a standard feature so that you can integrate it into your network as a client.

The control transmits data via the Ethernet card using the following protocols:

- the **smb** protocol (Server Message Block) for Windows operating systems, or
- The **TCP/IP** protocol family (**T**ransmission **C**ontrol **P**rotocol/**I**nternet **P**rotocol) and with support from the NFS (**N**etwork **F**ile **S**ystem)

> **i** Protect your data and your control by running your machines in a secure network.
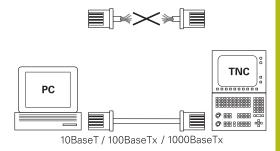
Connection possibilities

You can connect the Ethernet card in your control to your network through the RJ45 connection (X26,1000BaseTX, 100BaseTX and 10BaseT), or directly to a PC. The connection is metallically isolated from the control electronics.

For a 1000Base TX, 100BaseTX, and 10BaseT connection, use a twisted-pair cable to connect the control to your network.

> **i** The maximum possible cable length depends on the quality grade of the cable, the sheathing, and the type of network (1000BaseTX, 100BaseTX, or 10BaseT)

10BaseT / 100BaseTx / 1000BaseTx

General network settings

> **i** Have a network specialist configure the control.

Proceed as follows to access the general network settings:

| MOD | ▶ Press the **MOD** key |

▶ Enter the code number **NET123**

| PGM MGT | ▶ Press the **PGM MGT** key |

| NET | ▶ Press the **NET** soft key |

| CONFIGURE NETWORK | ▶ Press the **CONFIGURE NETWORK** soft key |
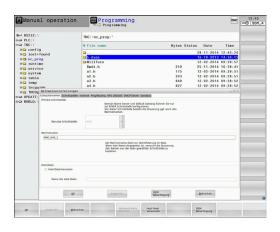
Computer name tab

ℹ	This setup dialog is managed by the HEROS operating system. After you have changed the conversational language on your control, you need to restart the control to enable the language.

Setting	Meaning
Primary interface	Name of the Ethernet interface to be integrated in your company network. Only active if a second, optional Ethernet interface is available on the control hardware
Computer name	Name displayed for the control in your company network
Host file	**Only required for special applications**: Name of a file in which the assignments of IP addresses to computer names are defined

Interfaces tab
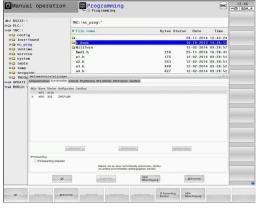
Setting	Meaning
Interface list	List of the active Ethernet interfaces. Select one of the listed interfaces (via mouse or arrow keys)
	▪ **Activate** button: Activate the selected interface (**X** appears in the **Active** column)
	▪ **Deactivate** button: Deactivate the selected interface (**-** appears in the **Active** column)
	▪ **Configuration** button: Open the configuration menu
Allow IP forwarding	**This function must be kept deactivated**. Only activate this function after instruction by our Service Department for diagnostic purposes. If external access to the second, optional Ethernet interface is necessary, this function needs to be activated.

Proceed as follows to access the configuration menu:
▶ Click the **Configuration** button

Setting	Meaning
Status	▪ **Interface active**: Connection status of the selected Ethernet interface
	▪ **Name**: Name of the interface you are currently configuring
	▪ **Plug connection:** Number of the plug connection of this interface on the logic unit of the control
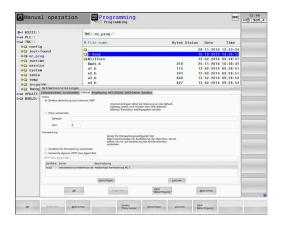
Setting	Meaning
Profile	Here you can create or select a profile in which all settings shown in this window are stored. HEIDENHAIN provides two standard profiles: ■ **DHCP-LAN**: Settings for the standard Ethernet interface; should work in a standard company network ■ **MachineNet**: Settings for the second, optional Ethernet interface; for configuration of the machine network Press the corresponding buttons to save, load and delete profiles
IP address	■ **Automatically procure IP address** option: The control is to procure the IP address from the DHCP server ■ **Manually set IP address** option: Manually define the IP address and subnet mask. Input: Four numerical values separated by periods, e.g.**160.1.180.20** and **255.255.0.0**
Domain Name Server (DNS)	■ **Automatically procure DNS** option: The control is to automatically procure the IP address of the domain name server ■ Option **Manually configure the DNS**: Manually enter the IP addresses of the servers and the domain name
Default gateway	■ **Automatically procure default gateway** option: The control is to automatically procure the default gateway ■ **Manually configure the default gateway** option: Manually enter the IP addresses of the default gateway

▶ Apply the changes with the **OK** button, or discard them with the **Cancel** button

Internet tab
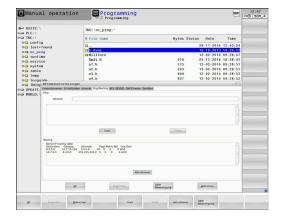
Setting	Meaning
Proxy	■ **Direct connection to Internet / NAT**: The control forwards Internet inquiries to the default gateway and from there they must be forwarded through network address translation (e.g. if a direct connection to a modem is available) ■ **Use proxy**: Define the **Address** and **Port** of the Internet router in your network, ask your network administrator for the correct address and port
Telemaintenance	The machine manufacturer configures the server for telemaintenance here. Changes must always be made in agreement with your machine tool builder

Ping/Routing tab

Setting	Meaning
Ping	In the **Address:** input field, enter the IP number for which you want to check the network connection. Input: Four numerical values separated by periods, e.g.**160.1.180.20**. As an alternative, you can enter the name of the computer whose connection you want to check ■ Press the **Start** button to begin the test. The control shows the status information in the Ping field ■ Press the **Stop** button to conclude the test
Routing	For network specialists: Status information of the operating system for the current routing ■ Press the **Update** button to refresh the routing information

NFS UID/GID tab

ℹ️ If user administration is active, the control will not display this tab. You will find the user-specific settings in user administration.
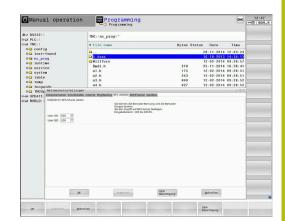
On the **NFS UID/GID** tab, you enter the user and group identifications.

Setting	Meaning
Set UID/GID for NFS shares	■ **User ID**: Definition of which user identification the end user uses to access files in the network. Ask your network specialist for the proper value ■ **Group ID**: Definition of the group identification with which you access files in the network. Ask your network specialist for the proper value
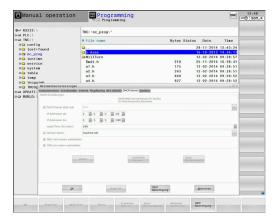
DHCP Server tab

Setting	Meaning
DHCP server	■ **IP addresses from**: Define the IP address as of which the control is to derive the pool of dynamic IP addresses. The control transfers the values that appear dimmed from the static IP address of the defined Ethernet interface; these values cannot be edited. ■ **IP addresses to**: Define the IP address up to which the control is to derive the pool of dynamic IP addresses. ■ **Lease Time (hours)**: Time within which the dynamic IP address is to remain reserved for a client. If a client logs on within this time, the control reassigns the same dynamic IP address. ■ **Domain name**: Here you can define a name for the machine network if required. This is necessary if the same names are assigned in the machine network and in the external network, for example. ■ **Forward DNS to external**: If **IP Forwarding** is active (Interfaces tab) and the option is active, you can specify that the name resolution for devices in the machine network can also be used by the external network. ■ **Forward DNS from external**: If **IP Forwarding** is active (Interfaces tab) and the option is active, you can specify that the control is to forward DNS inquiries from devices within the machine network to the name server of the external network if the DNS server of the MC cannot answer the inquiry. ■ **Status** button: Call an overview of the devices that are provided with a dynamic IP address in the machine network. You can also select settings for these devices. ■ **Advanced options** button: Additional settings for the DNS/DHCP server. ■ **Set standard values** button: Set factory default settings.

Sandbox tab

On the **Sandbox** tab, you configure the sandbox feature.

With the sandbox feature, the control enables you to execute applications in an environment that is isolated from the rest of the control. By isolating the data accesses, applications that run within a sandbox container have no access to files outside the virtual environment. This can be used, for example, to run a browser with access to the Internet.

> **i** Configure and use the sandbox on your control. For safety and security reasons, always open the browser in the sandbox.

Proceed as follows to activate the sandbox:

▶ Activate the Sandbox option (by selecting it)
> The control activates the default settings for the sandbox.
> When the default settings are active, the control offers you to start the browser from within the sandbox.

The sandbox can share a network connection (e.g. eth0) with the control. You can use the **Configure** button to configure separate network settings for the sandbox.

> **i** You can define firewall settings for the sandbox using the **brsb0** interface.
> **Further information:** "Firewall", Page 476

With the aid of the network settings, this enables you to allow only the sandbox to access the Internet. The control is only allowed to access its local Intranet or machine network. In this case, the browser is only allowed to access the Internet if the browser runs within the sandbox.

The sandbox is automatically given its own computer name. The computer name of the control is therefore suffixed with **_sandbox**.

Device-specific network settings

> 🛈 Have a network specialist configure the control.

You can define any number of network settings, but you can manage only seven at one time.

Proceed as follows to access the device-specific network settings:

| MOD | ▶ Press the **MOD** key |

Alternative:

| PGM MGT | ▶ Press the **PGM MGT** key |

| NET | ▶ Press the **NET** soft key |

| DEFINE NETWORK CONNECTN. | ▶ Press the **DEFINE NETWORK CONNECTN.** soft key |
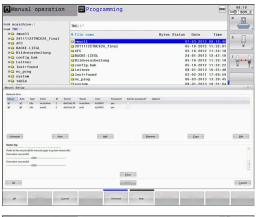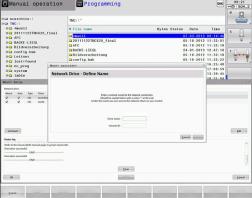
To manage the network drives, use the screen buttons.

Proceed as follows to add a network drive:

▶ Press the **Add** button

> The control starts the connection wizard, which guides you by dialog through the required definitions.

Setting	Meaning
Network drive	List of all connected network drives. The control shows the respective status of the network connections in the columns:
	■ **Mount**: Network drive connected / not connected
	■ **Auto**: Network drive is to be connected automatically/manually
	■ **Type**: Type of network connection. cifs and nfs are possible
	■ **Drive**: Designation of the drive on the control
	■ **ID**: Internal ID that identifies if a mount point has been used for more than one connection
	■ **Server**: Name of the server
	■ **Share**: Name of the directory on the server that the control is to access
	■ **User**: User name with which the user logs on to the network
	■ **Password**: Network password protected or not
	■ **Query password?**: Query / do not query password during connection
	■ **Options**: Display additional connection options
Status log	Display of status information and error messages.
	Press the Clear button to delete the contents of the Status Log window.

12.7 SELinux security software

SELinux is an extension for Linux-based operating systems. SELinux is an additional security software package based on Mandatory Access Control (MAC) and protects the system against the running of unauthorized processes or functions and therefore protects against viruses and other malware.

MAC means that every action must be explicitly permitted, otherwise it will not be executed by the control. The software is intended as protection in addition to the normal access restriction in Linux. Certain processes and actions can only be executed if the standard functions and access control of SELinux permit it.

> **i** The SELinux installation of the control has been prepared to permit running only programs installed with the HEIDENHAIN NC software. Other programs cannot be run with the standard installation.

The access control of SELinux under HEROS 5 is regulated as follows:

- The control executes only applications that are installed with the HEIDENHAIN NC software
- Files in connection with the security of the software (SELinux system files, HEROS 5 boot files, etc.), may only be modified by programs that have been selected explicitly.
- New files generated by other programs must never be executed
- USB data carriers cannot be deselected
- There are only two processes that are permitted to execute new files:
    - Starting a software update: A software update from HEIDENHAIN can replace or change system files
    - Starting the SELinux configuration: The configuration of SELinux is usually password-protected by your machine manufacturer; refer here to the relevant machine manual

> **i** HEIDENHAIN recommends activating SELinux because it provides additional protection against attacks from outside.

12.8 User administration

Introduction

⚙ Refer to your machine manual!
Some user administration areas are configured by the machine tool builder.

User administration is inactive in the control's factory default setting. This status is called **legacy mode**. In **legacy mode**, the control's behavior is identical to that of older software versions without user administration.

It is not mandatory that you use the user administration feature, but it is essential for the implementation of an IT security system.

User administration supports you in the following fields of security, based on the requirements of the IEC 62443 series of standards:

- Application security
- Network security
- Platform security

User administration enables you to define users with different access rights.

The following options are available for saving your user data:

- **Local LDAP database**
  - Using the user administration feature on one single control
  - Setting up a central LDAP server for more than one control
  - Exporting an LDAP server configuration file if the exported database is to be used by more than one control
    Further information: "Local LDAP database", Page 500
- **LDAP on remote computer**
  - Importing an LDAP server configuration file
    Further information: "LDAP on remote computer", Page 500
- **Connection to Windows domain**
  - Integrating the user administration feature on more than one control
  - Using different roles on different controls
    Further information: "Connection to a Windows domain", Page 501

ℹ Parallel operation of Windows users and users from an LDAP database is possible.

Configuring the user administration

User administration is inactive in the control's factory default setting. This status is called **legacy mode**.

Before user administration can be used, you need to configure it.

Perform the following steps for configuration:

1. Activate the user administration and create the **useradmin** user

2. Set up a database

- **Further information:** "Local LDAP database", Page 500
- **Further information:** "LDAP on remote computer", Page 500
- **Further information:** "Connection to a Windows domain", Page 501

3. Creating further users

Further information: "Creating further users", Page 504

Calling user administration

Proceed as follows to call user administration:

▶ Select the **HEROS** menu icon

▶ Select the **Settings** menu item

▶ Select the **UserAdmin** menu item

> The control opens the **User management** window.
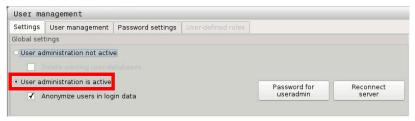
ℹ You can exit the **User management** window after each configuration step.

If you exit the **User management** window directly after having activated user administration, the control will prompt you for a restart once.

Enabling user administration

Proceed as follows to activate the user administration:

▶ Call the user administration

▶ Press the **User administration active** soft key

> The control shows the message **Password for user 'useradmin' missing**.

ℹ The purpose of the **Anonymize users in login data** function is data privacy; this function is active by default. While this function is active, user data in all log files of the control will be anonymized.

NOTICE

Caution: Unwanted data transfer is possible!

If you deactivate the **Anonymize users in login data** function, the system will show personalized user data in all control log files.
If servicing becomes necessary or if the log files need to be transmitted for another reason, the contracting party will be able to view this user data. In this case, it is your responsibility to ensure that any data protection provisions have been made at your company.

In order to prevent unwanted data transfers, make sure that the **Anonymize users in login data** function remains activated or reactivate it.

Deactivating user administration

User administration can only be deactivated by the following function users:

- **useradmin**
- **OEM**
- **SYS**

Further information: "HEIDENHAIN function users", Page 509

To deactivate user administration, proceed as follows:

▶ Log in as the appropriate function user
▶ Calling user administration
▶ Select **User administration inactive**

APPLY

▶ Press the **APPLY** soft key

END

▶ Press the **END** soft key

> The control opens the **System reboot required** window.
▶ Select **Yes**
> The control triggers a restart.

Create useradmin function user

After user administration has been activated, create the **useradmin** function user.

Proceed as follows to create the **useradmin** user:

▶ Select **Password for useradmin**
> The control opens the **Password for user 'useradmin'** pop-up window.
▶ Set the password for the **useradmin** user
▶ Select **Set new password**
> The message **Settings and password for 'useradmin' were changed** appears.

> ℹ For reasons of security, passwords should comply with the following criteria:
> - Eight characters minimum
> - Letters, numbers and special characters
> - Avoid using whole words or a sequence of characters, e.g. Anna or 123

The **useradmin** user is comparable to the local administrator of a Windows system.

The **useradmin** account provides the following functions:

- Creating databases
- Assigning the password data
- Activating the LDAP database
- Exporting LDAP server configuration files
- Importing LDAP server configuration files
- Emergency access if the user database was destroyed
- Retroactive change of the database connection
- Deactivating the user administration
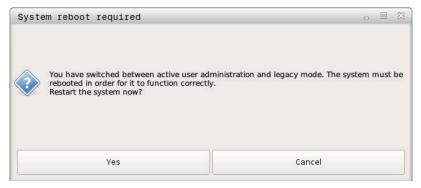
ℹ The **useradmin** user automatically receives the **HEROS.Admin** role. This allows him to manage users in the user administration, provided that he knows the LDAP database password. The **useradmin** user is a function user who has been predefined by HEIDENHAIN. You can neither add roles to function users, nor delete their roles.

Further information: "Definition of roles", Page 510

HEIDENHAIN recommends permitting more than one person to access an account with the **HEROS.Admin** role. This ensures that necessary changes to user administration can also be made in the administrator's absence.

Set up a database

Proceed as follows to set up the database:

- ▶ Select the database for saving the user data
- ▶ Set up a database
- ▶ Press the **APPLY** soft key
- ▶ Press the **END** soft key
- > The control opens the **System reboot required** window.
- ▶ Press **Yes** to restart the system
- > The control restarts.

Local LDAP database

Before you can use the **Local LDAP database** function, the following requirements must be met:

- The user administration is active
- The **useradmin** user has been configured

To set up a **Local LDAP database**, proceed as follows:

▶ Call the user administration

▶ Select the **LDAP user database** function

> The control enables the dimmed area for editing the LDAP user database.

▶ Select the **Local LDAP database** function

▶ Select the **Configuration** function

> The control opens the **Configure local LDAP database** window.

▶ Enter the name of the **LDAP domain**

▶ Enter password

▶ Repeat password

▶ Press the **OK** soft key

> The control closes the **Configure local LDAP database** window.

```
Configure local LDAP database                    _ □ ×
There is already a local LDAP domain. If you change the domain name, all
user entries will be lost.

Name of the LDAP domain:      HEROS5-LDAP            Reset
                                                    The server name is the same as
Name of the server:           DE01PC15451-340595    the hostname and can only be
                                                    modified in the network
Password:                                           configuration.



   OK        Export      Import      Cancel
             user data   user data
```

> [!INFO]
> Before you can start editing the user administration, the control prompts you to enter the password of your local LDAP database.
>
> Passwords must not be trivial and must be known only to the administrators.
>
> **Further information:** "Creating further users", Page 504
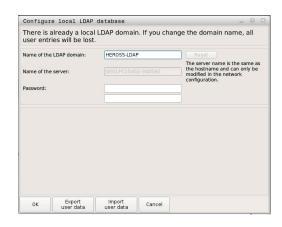
> [!INFO]
> If the host name or domain name of the control changes, you need to reconfigure the local LDAP databases.

LDAP on remote computer

Requirements

Before you can use the **LDAP on remote computer** function, the following requirements must be met:

- The user administration is active
- The **useradmin** user has been configured
- An LDAP database has been set up in the company network
- A server configuration file of an existing LDAP database must be stored on the control or on a PC in the network
- The PC with the existing configuration file is running.
- The PC with the existing configuration file is accessible in the network

Providing a server configuration file

To provide a server configuration file for an LDAP database, proceed as follows:

▶ Call the user administration

▶ Select the **LDAP user database** function

> The control enables the dimmed area for editing the LDAP user database.

▶ Select the **Local LDAP database** function

▶ Select the **Export server configuration** function

> The control opens the **Export LDAP configuration file window.**

▶ Enter the name for the server configuration file into the name field

▶ Save the file to the desired folder

> The server configuration file was exported successfully.

Using the LDAP database on another computer

To use the **LDAP on remote computer** function, proceed as follows:
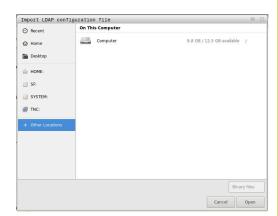
▶ Call the user administration

▶ Select the **LDAP user database** function

> The control enables the dimmed area for editing the LDAP user database.

▶ Select the **LDAP on remote computer** function

▶ Select the **Import server configuration** function

> The control opens the **Import LDAP configuration file window.**

▶ Select the existing configuration file

▶ Select **FILE**

▶ Press the **APPLY** soft key

> The configuration file was imported.

Connection to a Windows domain

Prerequisites

Before you can use the **Connection to Windows domain** function, the following requirements must be met:

■ User administration is active

■ The **useradmin** user has already been configured

■ There is a Windows Active Domain Controller in the network

■ You have access to the password of the domain controller

■ You have access to the user interface of the domain controller or you are supported by an IT administrator

■ The domain controller is accessible in the network

Setting up the Connection to Windows domain function

To set up the **Connection to Windows domain** function, proceed as follows:

▶ Call the user administration

▶ Select the **Connection to Windows domain** function

▶ Select the **Find domain** function

> The control recognizes the found domain.

> 🛈 Use the **Configuration** function to define various settings of your connection:
>
> ■ Deactivate the **Map SIDs to Unix UIDs** function
> ■ You can define a special group of Windows users to whom you want to restrict the connection to this control
> ■ You can modify the organizational unit in which the HEROS role names are stored
> ■ You can change the prefix in order to manage users from different workshops, for example. Each prefix given to a HEROS role name can be changed, e.g. HEROS hall 1 and HEROS hall 2
> ■ You can change the separator within the HEROS role name

▶ Press the **APPLY** soft key

> The control opens the **Connect to domain** window.

> 🛈 With the **Organizational unit for computer account:** function, you can specify in which of the already existing organizational units you want to create the access, e.g.
>
> ■ ou = controls
> ■ cn = computers
>
> The values you enter must match the conditions of the domain. The terms are not exchangeable.

▶ Enter the user name of the domain controller

▶ Enter the password of the domain controller

> The control connects to the Windows domain found.

> The control checks whether all of the required roles have been created in the domain as groups.

![i]	If not all of the required roles have been created in the domain as groups, the control issues a warning.
	If the control issues a warning, proceed in one of the two following ways:
	▶ Press the **Add role definition** soft key
	■ Select the **Add** function
	With this function, you can enter the roles directly for the domain.
	■ Select the **Export** function
	With this function, you can export the roles to a file in .ldif format.

> All of the required roles have been created in the domain as groups.

☐ Connection to Windows domain			
Domain: KDC:		Configuration	Find domain
LDAP ID-mapping: Yes			
HEROS role base:			Add role definition
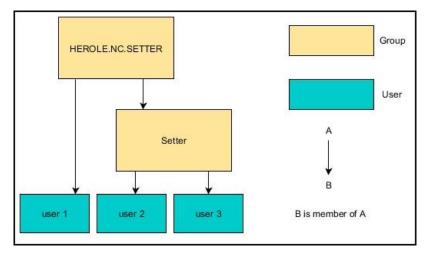
There are the following ways to create groups corresponding to the different roles:
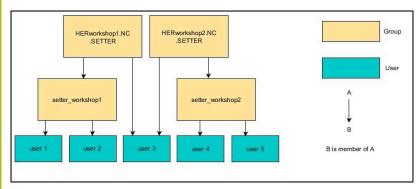
■ Automatically when entering the Windows domain by specifying a user with administrator rights
■ By importing an import file in .ldif format to the Windows server

The Windows administrator must add the users manually to the roles (security groups) on the domain controller.

Two suggestions describing how the groups can be structured by the Windows administrator are given by HEIDENHAIN below:

■ Suggestion 1: The user is a direct or indirect member of the respective group:

■ Suggestion 2: Users from various sectors (workshops) are
members of groups with different prefixes:

Creating further users

After configuring user administration, you can create further users.

Before you can create further users, you must have configured and
selected an LDAP database.

Proceed as follows to create further users:

▶ Call the user administration

▶ Select the **User management** tab

> 🛈 The **User management** tab is only effective for the
> following databases:
> ■ **Local LDAP database**
> ■ **LDAP on remote computer**
> If **Connection to Windows domain** is used, you
> need to configure the users in the Windows
> domain.
> **Further information:** "Connection to a Windows
> domain", Page 501
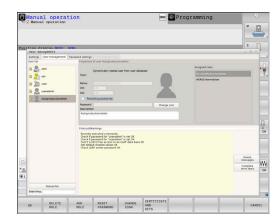
▶ Press the **EDIT ON** soft key

> The control prompts you to enter the password of your user
> database.

> 🛈 This step is only needed if you have restarted your
> control after configuring the database.

> After you have entered the password, the control opens the
> **User management** menu.

You can edit the existing users or create new users.

Proceed as follows to create a new user:

▶ Press the **Create new user** soft key
> The control opens a window for creating a user.
▶ Enter the user name
▶ Enter a password for the user

> **ℹ** The user must change the password when logging in for the first time.
>
> **Further information:** "Login to user administration", Page 517

▶ You can optionally also create a user description
▶ Press the **Add role** soft key
▶ In the selection window, select the roles corresponding to your user
Further information: "Definition of roles", Page 510
▶ Confirm your selection with the **ADD** soft key

> **ℹ** Two additional soft keys are available in the menu:
> ▪ **ADD REMOTE LOGIN:**
>
> Adds **Remote.HEROS.Admin** in place of **HEROS.Admin**, for example.
>
> This role is only enabled for remote logon to the system.
> ▪ **ADD LOCAL LOGIN**
>
> Adds **Local.HEROS.Admin** in place of **HEROS.Admin**, for example.
>
> This role is only enabled for local logon at the control's screen.

▶ Press the **CLOSE** soft key
> The control closes the window for creating a user.
> Press the **OK** soft key
▶ Press the **APPLY** soft key
> The control adopts the changes.
▶ Press the **END** soft key
> The control closes the user administration.

> **ℹ** If you have not restarted your control after configuring the database, the control prompts you to restart for the changes to take effect.
>
> **Further information:** "Configuring the user administration", Page 496

Adding profile images

You can optionally also assign images to your users. The **Default user icons:** from HEIDENHAIN are available for this purpose.
You can also load your own images in JPEG or PNG format to the control. Then you can use these image files as profile images.

Proceed as follows to define profile images:

▶ Log on the user with **HEROS.Admin** role, e.g. **useradmin**

Further information: "Login to user administration", Page 517

▶ Call the user administration

▶ Select the **User management** tab

▶ Press the **Edit user** soft key

▶ Press the **Change icon** soft key

▶ Select the desired image in the menu

▶ Press the **Choose icon** soft key

▶ Press the **OK** soft key

▶ Press the **APPLY** soft key

> The control adopts the changes.

ℹ You can also directly add profile images when creating the users.

Password settings in user administration

Password settings tab

Users with the **HEROS.Admin** role can specify exact requirements for user passwords on the **Password settings** tab.
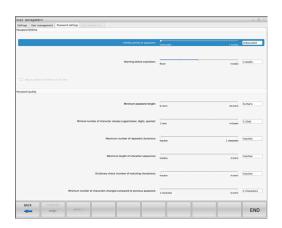
Further information: "Privileges", Page 513

ℹ If you do not comply with the requirements defined here when creating your password, the control will issue an error message.

To open the **Password settings** tab, proceed as follows:

▶ Log on as a user with **HEROS.Admin** role

▶ Calling user administration

▶ Select the **Password settings** tab

▶ Press the **EDIT ON** soft key

> The control opens the **Enter password of LDAP database** window

▶ Enter password

> The control releases the **Password settings** tab for editing.

Defining password settings

The control allows you to configure user password requirements by setting various parameters.

To edit the parameters, proceed as follows:

▶ Open the **Password settings** tab
▶ Select the desired parameter
> The control highlights the selected parameter in blue.
▶ Set the desired parameter using the slider.
> The control shows the selected parameter in the display field.

APPLY	▶ Press the **UNIT DATA** soft key
	> The control applies the change.

The following parameters are available:

Password lifetime

■ **Validity period of password:**

Here, you can indicate how long the password can be used.

■ **Warning before expiration:**

From the defined time, a warning will be issued that the password will soon expire.

Password quality

■ **Minimum password length:**

Here, you can indicate the minimum password length.

■ **Minimal number of character classes (upper/lower, digits, special):**

Here, you can indicate the minimum number of different character classes required in the password.

■ **Maximum number of repeated characters:**

Here, you can indicate the maximum number of identical successive characters in the password.

■ **Maximum length of character sequences:**

Here, you can indicate the maximum length of the character sequences to be used in the password, e.g. 123.

■ **Dictionary check (number of matching characters):**
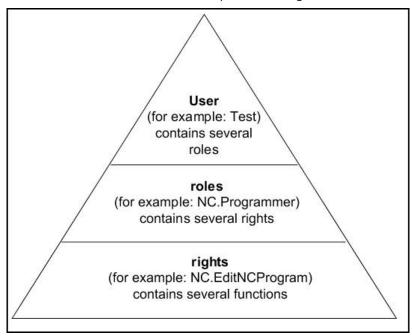
Here, you can enable a check whether the password contains known words and specify the allowed number of meaningful characters.

■ **Minimum number of characters changed compared to previous password:**

Here, you can specify how many characters in the new password must be different from the previous one.

Access rights

The user administration is based on the Unix rights management.
Access to the control is controlled by means of rights.

The user administration differentiates between the following terms:

- **User**
  - Function users predefined by HEIDENHAIN
    Further information: "HEIDENHAIN function users",
    Page 509
  - Function user defined by the machine tool builder
  - Self-defined users

    A user can be pre-defined in the control or he can be defined
    by the end user. A user contains all the roles that have been
    assigned to him.

> **i**   Your machine tool builder defines function users
> who are required for specific tasks such as machine
> maintenance.
>
> Depending on the task assigned, you can use one of
> the pre-defined function users or you have to create a
> new user.
>
> The access rights of HEIDENHAIN function users are
> already pre-defined in the control's factory default
> setting.

Roles

Roles consist of a combination of rights that cover certain
functions provided by the control.

- **Operating system roles:**
- **NC operator roles:**
- **Machine tool builder (PLC) roles:**

All of these roles are pre-defined in the control.

You can assign more than one role to a user.

Privileges

Rights consist of a combination of functions that cover a task area of the control, e.g. editing the tool table.

- HEROS rights
- NC rights
- PLC rights (OEM)

If more than one role is assigned to a user, he will be granted all rights contained in these roles.

> ⓘ   Ensure that every user is assigned all access rights he needs. The access rights result from the actions a user performs on the control.

HEIDENHAIN function users

HEIDENHAIN function users are pre-defined users that are automatically created upon activation of user administration. Function users cannot be changed.

HEIDENHAIN provides four different function users in the control's factory default setting.

- **oem**

  The **oem** function user is intended for the machine tool builder. With **oem**, you can access the PLC partition of the control.

- **Function user defined by the machine tool builder**

  > ⚙   Refer to your machine manual!
  >
  > The users defined by the machine tool builder can differ from those pre-defined by HEIDENHAIN.
  >
  > The machine tool builder's function users can already be active in **legacy mode** and replace code numbers.
  >
  > By entering code numbers or passwords that replace code numbers, you can temporarily enable rights of **oem** function users.
  >
  > **Further information:** "Current User", Page 524

- **sys**

  With the **sys** function user, you can access the system partition of the control. This function user is reserved for use by the JH service personnel.

- **user**

  In **legacy mode**, the **user** function user is automatically logged on to the system during control start-up. When user administration is active, the **user** function user has no effect. The logged-on user of the type **user** cannot be changed in **legacy mode**.

- **useradmin**

  The **useradmin** function user is automatically created upon activation of user administration. With **useradmin**, you can configure and edit user administration.

Definition of roles

HEIDENHAIN combines several rights for individual task areas to roles. Different pre-defined roles that you can use to assign rights to your users are available. The tables below describe the individual rights of the different roles.

> **i** Every user should have at least one role from the operating system area and at least one role from the programming area.
>
> You can enable a role either for local login or for remote login. With local login, the user directly logs on to the control at the control's screen. A remote login (DNC) is a connection via SSH.
>
> You can therefore also make the rights of a user dependent on the access used to operate the control.
>
> If a role is only enabled for local login, **Local.** is added to the role name, e.g. **Local.HEROS.Admin** instead of **HEROS.Admin**.
>
> If a role is only enabled for remote login, **Remote.** is added to the role name, e.g. **Remote.HEROS.Admin** instead of **HEROS.Admin**.

Advantages of classification in roles:

- Simplified administration
- Different rights are compatible between different software versions of the control and different machine manufacturers.

> **i** Various applications require access to certain interfaces. Where required, the administrator also needs to set up rights to access the required interfaces as well as rights to access certain functions and additional programs. These rights are granted by the **Operating system roles:**.

> **i** The following contents can change in the following software versions of the control:
> - HEROS role names
> - Unix groups
> - Basic ID number

Operating system roles:

Role	Privileges		
	HEROS role name	**UNIX group**	**Basic ID number**
HEROS.RestrictedUser	Role for a user with minimum rights on the operating system.		
	■ HEROS.MountShares	■ mnt	■ 332
	■ HEROS.Printer	■ lp	■ 9
HEROS.NormalUser	Role for a normal user with limited rights on the operating system.		
	This role grants the rights of the **RestrictedUser** role, as well as the following rights:		
	■ HEROS.SetShares	■ mntcfg	■ 331
	■ HEROS.ControlFunctions	■ ctrlfct	■ 337
HEROS.LegacyUser	With the **legacy user** role, the behavior regarding the operating system of the control is identical to that of older software versions without user administration. The user administration remains active.		
	This role grants the rights of the **NormalUser** role, as well as the following rights:		
	■ HEROS.BackupUsers	■ userbck	■ 334
	■ HEROS.PrinterAdmin	■ lpadmin	■ 16
	■ HEROS.SWUpdate	■ swupdate	■ 338
	■ HEROS.SetNetwork	■ netadmin	■ 333
	■ HEROS.SetTimezone	■ tz	■ 330
	■ HEROS.VMSharedFolders	■ vboxsf	■ 1000
HEROS.Admin	The configuration of the network and the configuration of the user administration are some of the rights granted by this role.		
	This role grants the rights of the **LegacyUser** role, as well as the following rights:		
	■ HEROS.UserAdmin	■ useradmin	■ 336

NC operator roles:

Role	Privileges		
	HEROS role name	**UNIX group**	**Basic ID number**
NC.Operator	This role allows you to run NC programs.		
	▪ NC.OPModeProgramRun	▪ NCOpPgmRun	▪ 302
NC.Programmer	This role grants the rights of NC programming.		
	This role grants the rights of the **Operator** role, as well as the following rights:		
	▪ NC.EditNCProgram	▪ NCEdNCProg	▪ 305
	▪ NC.EditPalletTable	▪ NCEdPal	▪ 309
	▪ NC.EditPresetTable	▪ NCEdPreset	▪ 308
	▪ NC.EditToolTable	▪ NCEdTool	▪ 306
	▪ NC.OPModeMDi	▪ NCOpMDI	▪ 301
	▪ NC.OPModeManual	▪ NCOpManual	▪ 300
NC.Setter	This role allows you to edit the pocket table.		
	This role grants the rights of the **Programmer** role, as well as the following rights:		
	▪ NC.ApproveFsAxis	▪ NCApproveFsAxis	▪ 319
	▪ NC.EditPocketTable	▪ NCEdPocket	▪ 307
	▪ NC.SetupDrive	▪ NCSetupDrv	▪ 315
	▪ NC.SetupProgramRun	▪ NCSetupPgRun	▪ 303
NC.AutoProductionSetter	This role allows you to execute all NC functions, including programming a scheduled NC program start.		
	This role grants the rights of the **Setter** role, as well as the following rights:		
	▪ NC.ScheduleProgramRun	▪ NCSchedulePgRun ▪ 304	
NC.LegacyUser	With the **legacy user** role, the control's behavior regarding NC programming is identical to that of older software versions without user administration. User administration remains active. The **legacy user** has the same rights as the **AutoProductionSetter.**		
NC.AdvancedEdit	This role allows you to use special functions of the NC and table editors. ▪ **FN 17** and editing the table head Replacement for code number **555343**		
	▪ NC.EditNCProgramAdv	▪ NCEditNCPgmAdv ▪ 327	
	▪ NC.EditTableAdv	▪ NCEditTableAdv ▪ 328	
NC.RemoteOperator	This role allows to start NC programs from an external application.		
	▪ NC.RemoteProgramRun	▪ NCRemotePgmRun ▪ 329	

Machine tool builder (PLC) roles:

Role	Privileges		
	HEROS role name	**UNIX group**	**Basic ID number**
PLC.ConfigureUser	This roles grants the rights on code number **123**.		
	■ NC.ConfigUserAdv ■ NC.SetupDrive	■ NCConfigUserAdv ■ NCSetupDrv	■ 316 ■ 315
PLC.ServiceRead	This role allows read-only access during servicing. This role can be used to display various types of diagnostic information		
	■ NC.Data.AccessServiceRead	■ NCDAServiceRead	■ 324

> **ℹ** Refer to your machine manual!
>
> The machine tool builder can adapt the PLC roles.
>
> When the **Machine tool builder (PLC) roles:** are adapted by the machine tool builder, the following contents may change:
> - The names of the roles
> - The number of roles
> - The functionality of the roles

Privileges

The table below lists all of the individual rights.

Rights:

HEROS role name	Description
HEROS.Printer	Data output to network printers
HEROS.PrinterAdmin	Configuration of network printers
NC.OPModeManual	Operation of the machine in the **Manual Operation** and **Electronic handwheel** operating modes
NC.OPModeMDi	Working in the **Positioning w/ Manual Data Input** operating mode
NC.OPModeProgramRun	Execution of NC programs in **Program Run Full Sequence** or **Program run, single block** operating mode
NC.SetupProgramRun	Probing in the **Manual Operation** and **Electronic handwheel** operating modes
NC.ScheduleProgramRun	Program a scheduled NC program start
NC.EditNCProgram	Editing NC programs
NC.EditToolTable	Editing the tool table
NC.EditPocketTable	Edit Pocket Table
NC.EditPresetTable	Editing the preset table
NC.EditPalletTable	Editing pallet tables
NC.SetupDrive	Adjustment of drives by the end user
NC.ApproveFsAxis	Confirming test position of safe axes
NC.EditNCProgramAdv	Additional NC functions, e.g. **FN 17**
NC.EditTableAdv	Additional table programming functions, e.g. editing the table head

HEROS role name	Description
HEROS.SetTimezone	Adjustment of time and date, time zone, and time synchronization via NTP and HEROS menu
HEROS.SetShares	Configuration of network shares mounted by the control
HEROS.MountShares	Connecting and disconnecting network shares with the control
HEROS.SetNetwork	Configuration of network and relevant settings for data security
HEROS.BackupUsers	Data backup on the control—for all users configured on the control
HEROS.BackupMachine	Backup and restoring data of the entire machine configuration
HEROS.UserAdmin	Configuration of user administration on the control This includes creating, deleting and configuring local users
HEROS.ControlFunctions	Control function of the operating system ■ Auxiliary functions, such as starting and stopping NC software. ■ Telemaintenance ■ Advanced diagnostic functions, such as log data
HEROS.SWUpdate	Installation of software updates for the control
HEROS.VMSharedFolders	Access to shared folders of a virtual machine Only relevant when running a programming station within a virtual machine
NC.RemoteProgramRun	NC program start from an external application, e.g. via the DNC interface
NC.ConfigUserAdv	Configuration access to the contents that have been enabled through code number **123**
NC.Data.AccessServiceRead	Read-only access to the PLC partition during servicing

User authentication from external applications

Introduction

If user administration is active, external applications also need to authenticate a user so that the suitable rights can be assigned.

If LSV2 connections are used, the connection is routed via an SSH tunnel. This method assigns the remote user to a user set up on the control, granting the remote user this user's rights.

> **ℹ** Besides, the encryption used with the SSH tunnel protects the communication from attackers.

> **ℹ** For OPC UA connections, a stored user certificate is used for authentication.
> **Further information:** "HEIDENHAIN OPC UA NC Server (Options 56 - 61)", Page 526

Concept of transmission through an SSH tunnel
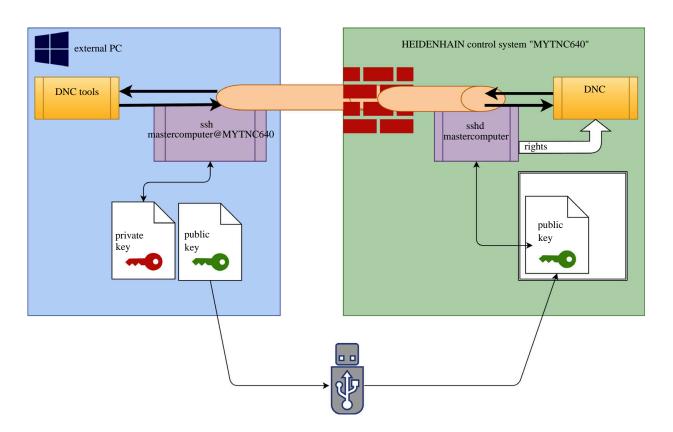
Prerequisites:

- TCP/IP network
- The remote computer acts as SSH client
- The control acts as SSH server
- Key pair consisting of
  - Private key
  - Public key

An SSH connection is always set up between an SSH client and an SSH server.

A key pair is used to protect the connection. This key pair is generated on the client. The key pair consists of a private key and a public key. The private key remains with the client. During setup, the public key is transferred to the server and assigned to a certain user.

The client tries to connect to the server using the pre-defined user name. The server can use the public key to verify that the requester of the connection holds the associated private key. If yes, the server accepts the SSH connection and assigns it to the user that has been used for the login. Communication can then be "tunneled" through this SSH connection.

Use in external applications

The PC tools available from HEIDENHAIN, such as TNCremo with version **v3.3** or higher, provide all functions for setting up, building up, and managing secure connections through an SSH tunnel.

When the connection is set up, the required key pair is generated in TNCremo and the public key is transferred to the control.

> **i** Once the connection configurations created in TNCremo have been set up, they can be shared among all PC tools for building up a connection.

This also applies to applications that are using the HEIDENHAIN DNC component from RemoTools SDK for communication. There is no need to adapt existing customer applications.

> **i** In order to expand the connection configuration using the associated **CreateConnections** tool, you need to update to **HEIDENHAIN DNC v1.7.1**. A modification of the application source code is not required.

Proceed as follows to set up a secure connection for the logged-on user:

▶ Select the **HEROS** menu item
▶ Select the **Settings** menu item
▶ Select the **Current User** menu item
▶ Press the **Certificate and keys** soft key
▶ Select the **Allow password authentication** function
▶ Press the **Store and restart server now** soft key
▶ Use the **TNCremo** application to set up the secure connection (TCP secure).

> **📖** For detailed information on how to proceed, please refer to the integrated help system of TNCremo.

> The public key has been stored on the control by TNCremo.

> **i** In order to ensure maximum security, the **Allow password authentication** function is deactivated after the public key has been stored.

▶ Deselect the **Allow password authentication** function
▶ Press the **Store and restart server now** soft key
> The control has adopted your changes.

> **i** In addition to using the PC tools for setup with password authentication, you can also import the public key into the control by using a USB stick or a network drive. However, this is not described in detail in this document.

Proceed as follows to delete a key in the control so that a user can no longer use the secure connection:

▶ Select the **HEROS** menu item
▶ Select the **Settings** menu item
▶ Select the **Current User** menu item
▶ Press the **Certificate and keys** soft key
▶ Select the key to be deleted
▶ Press the **Delete SSH key** soft key
> The control deletes the selected key.

Disabling insecure connections in the firewall

The DNC protocols LSV2 and RPC should be blocked in the firewall to ensure that the use of secure connections provides a real IT security advantage for the control.

This requires that the following parties change to secure connections:

■ Machine tool builders with all external applications, e.g. pick-and-place robots

> **ℹ** If the additional application is connected via **machine network X116**, there is no need to switch to an encrypted connection.

■ Users with custom external applications

If you have ensured that the connections of all parties are secure, the LSV2 and RPC DNC protocols can be blocked in the firewall.

To block the protocols in the firewall, proceed as follows:

▶ Select the **HEROS** menu item
▶ Select the **Settings** menu item
▶ Select the **Firewall** menu item
▶ Select the **Prohibit all** method for **DNC** and **LSV2**
▶ Select the **Apply** function
> The control saves your changes.
▶ Close the window with **OK**

Login to user administration

The login dialog appears in the following situations:
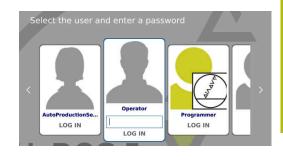
■ If user administration is active, the dialog appears immediately after the control has booted
■ After the **User logout** function has been executed
■ After the **Switch user** function has been executed
■ After the screen has been locked by the screensaver

The login dialog provides the following options:

■ Users who logged in at least once
■ **Other** user

To log on as a user who is already displayed in the login dialog, proceed as follows:

▶ Select the respective user in the login dialog
> The control enlarges the user icon you selected.

▶ Enter the user password

> The control uses the selected user to log you in.

To log on a user for the first time, you need to use the **Other** input field.

To log on as a user with the **Other** option for the first time, proceed as follows:

▶ Select **Other** in the login dialog

> The control enlarges the user icon you selected.

▶ Enter the user name

▶ Enter the user's password

> The control opens a window with the **Password expired** message. **Change the password now.**

▶ Enter the current password

▶ Enter the new password

▶ Repeat the new password

> The control uses the new user to log you in.

> The user is shown in the login dialog.

ℹ For reasons of security, passwords should comply with the following criteria:

■ Eight characters minimum

■ Letters, numbers and special characters

■ Avoid using whole words or a sequence of characters, e.g. Anna or 123

Keep in mind that an administrator can define the requirements to be met by a password. The password requirements include:

- Minimum length
- Minimum number of different types of characters
    - Capital letters
    - Small letters
    - Numbers
    - Special characters
- Maximum length of character strings, e.g. 54321 = string of five characters
- The number of characters must match the dictionary check
- Minimum number of characters changed compared to previous password

If the new password does not meet the requirements, an error message will be issued. This means that you have to enter a different password.

> **ⓘ** Administrators can specify the expiration of passwords. If you do not change your password within the valid period, you will no longer be able to log on as that user. In this case, an administrator needs to reset the user password before you can log on again.
>
> ▶ Change your password at regular intervals
>
> **Further information:** "Changing the current user's password", Page 525
>
> ▶ Take note of the password change warnings

Switching or logging out users

Use the **Switch-off** HEROS menu item or the icon of the same name at bottom right in the menu bar to open the **Switch off and restart** window.

The control provides the following options:

- **Switch-off**:
  - The control stops and ends all additional programs and functions
  - The system is shut down
  - The control is switched off
- **Restart**:
  - The control stops and ends all additional programs and functions
  - The system is restarted
- **Log out**:
  - The control ends all additional programs
  - The user is logged out
  - The login window appears

  > **i** To continue, you need to log in a new user and enter his password.
  >
  > The NC program continues running under the previously logged-on user.

- **Switch user**:
  - The login window appears
  - The user is not logged out

  > **i** You can close the login window with **Cancel** without entering a password.
  >
  > All additional programs and the NC programs started by the logged-on user continue running.
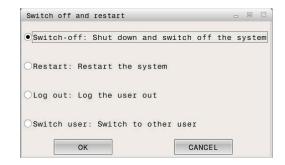
Screensaver with lockout
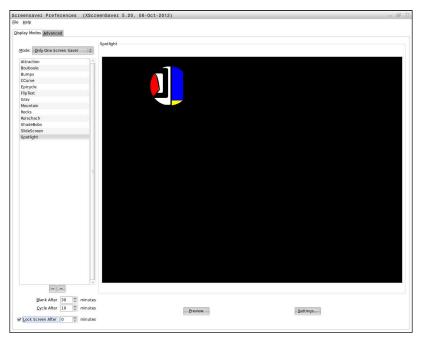
You can use the screensaver to lock the control. NC programs that have already been started continue running during this period.

> **i** You need to enter a password to unlock the screensaver again.
>
> **Further information:** "Login to user administration", Page 517

Proceed as follows to access the screensaver settings in the
HEROS menu:

▶ Select the **HEROS** menu item

▶ Select the **Settings** menu item

▶ Select the **Screensaver** menu item

The screensaver provides the following options:

◻ Use the **Blank After** setting to define after how many minutes
the screensaver should turn on.

◻ Use the **Lock Screen After** setting to activate the lock with
password protection.

◻ The time value behind **Lock screen after** defines the delay after
which the lock will take effect after the screensaver has been
turned on. A value of **0** means that the lock becomes effective
immediately after the screensaver has been turned on.

When the lock is active and you use one of the input devices
(e.g. the mouse), the screensaver disappears and the lock screen
appears.

Use **Unlock screen** or **Enter** to open the login window again.

HOME directory

When user administration is active, a private **HOME:** directory, to which you can save your private programs and files, is available to every user.

The **HOME:** directory can be viewed by any logged-on user.
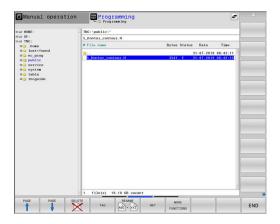
'public' directory

public **directory**

Upon the first activation of user administration, the **public** below the TNC partition will be connected.

The **public** directory can be accessed by any user.

Setting advanced file access rights

In order to control the use of individual files in the **public** directory, HEIDENHAIN provides the **ADVANCED ACCESS RIGHTS** function that allows you to restrict access on a file level.

Proceed as follows to call the **ADVANCED ACCESS RIGHTS** function:

▶ Select the **Programming** operating mode

PGM
MGT
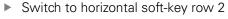
▶ Press the **PGM-MGT** key
▶ Switch to horizontal soft-key row 2

MORE
FUNCTIONS

▶ Press the **MORE FUNCTIONS** soft key
▶ Switch to horizontal soft-key row 2

ERWEITERTE
ZUGRIFFS-
RECHTE

▶ Press the **ADVANCED ACCESS RIGHTS** soft key
> The control opens the **Erweiterte Zugriffsrechte einstellen** window.

Setting file access rights

If you transfer files to the **public** directory or create files there, the control considers the logged-in user as the file owner. The owner can control access to his own files.

> **i** Access rights can only be defined for files located in the **public** directory.
>
> The control automatically assigns the **user** function user as the owner to any files stored on the TNC partition, but not in the **public** directory.

You can specify the access rights for the following users:

- **Owner:**

  The person who owns the file

- **Group:**

  A selected Linux group or users with a specific HEIDENHAIN access rights

- **Andere:**

  All users not belonging to the previously selected Linux group or not having the specified HEIDENHAIN access right.

You can set access rights as follows:

- **Read**

  The file can be viewed

- **Schreiben**

  The file can be edited

- **Ausführen**

  The file can be executed

Using the soft keys in the **Erweiterte Zugriffsrechte einstellen** window, you can select or deselect all access rights for the users:

ZUGRIFF BESITZER WECHSELN	▶ Select or deselect all access rights for the **Besitzer:**
ZUGRIFF GRUPPE WECHSELN	▶ Select or deselect all access rights for the **Group:**
ZUGRIFF ANDERE WECHSELN	▶ Select or deselect all access rights for **Andere:**
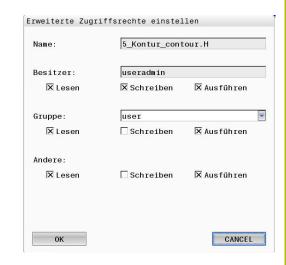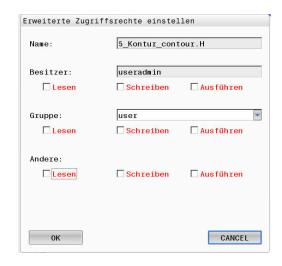
Proceed as follows to select the access rights for a group:

▶ Call the **ADVANCED ACCESS RIGHTS** function

▶ Select the desired group in the drop-down menu

▶ Select or deselect the access rights as required

> Any changes made are highlighted in red.

▶ Select **Ok**

> The changes made to access rights are applied.

Current User

With **Current User**, you can view the group rights of the currently logged-on user in the **HEROS** menu.

> **ⓘ** In legacy mode, the **user** function user is automatically logged on to the system during control start-up. When user administration is active, the **user** function user has no effect.
>
> **Further information:** "HEIDENHAIN function users", Page 509

Calling the **Current User**:

▶ Select the **HEROS** menu icon

▶ Select the **Settings** menu item

▶ Select the **Current User** menu icon

The user administration allows you to temporarily increase the current user's rights by adding the rights of a user you selected.

Proceed as follows to temporarily increase a user's rights:
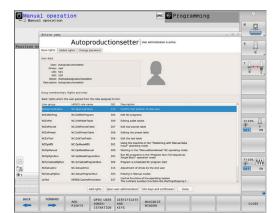
▶ Call **Current User**

▶ Press the **Add rights** soft key

▶ Select a user

▶ Enter the selected user's user name

▶ Enter the selected user's password

> The control temporarily increases the logged-on user's rights by adding the rights of the user entered in **Add rights**.

> **ⓘ** You can temporarily enable the rights of **oem** function users. For this purpose, enter the appropriate code number or the password defined by the machine tool builder.

You have the following options to remove the rights that have temporarily been added:

■ Enter the code number **0**

■ User logout

■ Press the **Remove added rights** soft key

Select the **Remove added rights** soft key as follows:

▶ Call **Current User**

▶ Select the **Added rights** tab

▶ Press the **Remove added rights** soft key

Changing the current user's password

In the **Current User** menu item, you can change your current user's password.

Proceed as follows to change your current user's password:

▶ Call **Current User**

▶ Select the **Change password** tab

▶ Enter the old password

▶ Press the **Verify old password** soft key

> The control checks whether you have correctly entered your old password.

> If the password is correct, the fields **New password** and **Repeat password** are enabled.

▶ Enter the new password

▶ Repeat the new password

▶ Press the **Set new password** soft key

> The control compares the requirements defined for passwords by the administrator to the password you entered.

Further information: "Login to user administration", Page 517

> The message **Password was changed successfully** appears.

Dialog for requesting additional rights

If you do not have the rights required for a specific HEROS menu item, the control opens the window for requesting additional rights:

In this window, you can temporarily obtain more rights by adding another user's rights.

In the **Users that have this right:** field, the control lists all existing users that have the right to this function.

ℹ️ If **Connection to Windows domain** is used, only users that were recently logged on are shown in the selection menu.

To attain the rights of users that are not shown, enter their user data. The control will then recognize those users that are contained in the user database.

Obtaining more rights

To temporarily add another user's rights to your rights, proceed as follows:

▶ Select the user whose rights you need

▶ Enter the user's name

▶ Enter the user's password

▶ Press the **Set right** soft key

> The control adds the entered user's rights to your rights.

Further information: "Current User", Page 524

12.9 HEIDENHAIN OPC UA NC Server (Options 56 - 61)

Introduction

Open **P**latform **C**ommunications **U**nified **A**rchitecture (OPC UA) refers to a collection of specifications. These specifications are used to standardize machine-to-machine communication (M2M) in the field of industrial automation. OPC UA enables the data exchange—across operating systems—between products from different manufacturers, e.g. between a HEIDENHAIN control system and third-party software. Thus, OPC UA has become the data exchange standard for a secure, reliable, manufacturer/platform-independent industrial communication over the last years.

For OPC UA-based communication, the HEIDENHAIN control features the **HEIDENHAIN OPC UA NC Server**. For each OPC UA client to be connected, you need one of the six available software options (56 - 61).

Both standard and custom software can be used with the **HEIDENHAIN OPC UA NC Server**. Compared to other established interfaces, significantly less development effort is required for OPC UA connection, thanks to the uniform communication technology.

The **HEIDENHAIN OPC UA NC Server** allows access to the data and functions of the HEIDENHAIN NC information model exposed in the server address space. The following OPC UA services are supported:

- Reading and writing variables
- Subscribing to value changes
- Executing methods
- Subscribing to events

Further information: "Application development", Page 529

IT Security

In 2016, the German Federal Office for Information Security (BSI) published a security analysis related to OPC UA. The specification analysis performed by the BSI revealed that OPC UA provides a high level of security, when compared to most other industrial protocols.

HEIDENHAIN follows the BSI recommendations and provides SignAndEncrypt, which exclusively features up-to-date IT security profiles. For this purpose, OPC UA-based industrial applications and the **HEIDENHAIN OPC UA NC Server** exchange certificates for authentication. In addition, any transferred data is encrypted. This effectively prevents messages between the communication partners from being intercepted or altered.

Among others, the HEROS **Connection Assistant** function supports you when setting up the certificates.

Further information: "Setting up the connection", Page 527

Machine configuration

Using the **HEIDENHAIN OPC UA NC Server**, OPC UA client applications can query general machine information, such as the year of construction of the machine or its location.

The following machine parameters are available for the digital identification of your machine:

- For users: **CfgMachineInfo** (no. 131700)
- For the machine tool builder: **CfgOemInfo** (no. 131600)

Setting up the connection

Easy configuration with the Connection Assistant

For quick and easy setup of an OPC UA client application, you can use the HEROS **Connection Assistant** function. This assistant guides through the steps that are required to connect an OPC UA client application to the control.

The assistant features the following steps:

- Export **OPC UA NC Server** certificates
- Import the certificates of the OPC UA client application
- Assign each of the available **OPC UA NC Server** software options to an OPC UA client application
- Import user certificates
- Assign user certificates to users
- Configure the firewall

> **i**   The **Connection Assistant** also helps you create test or sample certificates for users and the OPC UA client application Do not use the user and client application certificates created at the control for other purposes than development at the programming station.

Complex configuration with separate HEROS functions

Besides the simple setup with the **Connection Assistant**, the control provides separate HEROS functions for complex configurations:

■ **PKI Admin**

The **HEIDENHAIN OPC UA NC Server** is one of the applications whose **P**ublic **K**ey **I**nfrastructure (**PKI**) can be configured with the HEROS **PKI Admin** function. After launching the HEROS **PKI Admin** function and selecting the **OPC UA NC Server** application, you can use the extended functionality.

Further information: "PKI Admin", Page 530

■ **Current User** and **UserAdmin**

Each user of an OPC UA client application uses a certificate for authentication. For linking the certificates with users, the HEROS **Current User** or **UserAdmin** functions are available.

Further information: "User administration", Page 495

■ **OPC UA NC Server**

The HEROS **OPC UA NC Server** function includes a dialog named **License Settings** for managing the assignment of the active software options 56 to 61.

> **ⓘ** Before a certificate will be available for activation in the selection area of the **License Settings** dialog, you must import the corresponding certificate of a OPC UA client application with the HEROS **PKI Admin** function or the **Connection Assistant**.

■ **Firewall**

Make sure to configure your firewall accordingly so that the connection of OPC UA applications to the **OPC UA NC Server** will not be blocked.

Further information: "Firewall", Page 476

Application development

OPC UA is a manufacturer/platform-independent, open communication standard. For this reason, an OPC UA client SDK is not included in the **HEIDENHAIN OPC UA NC Server**.

HEIDENHAIN information model

A separate **Companion Specification** document is available that describes the information model supported by the **HEIDENHAIN OPC UA NC Server**.

⊕ The following sources provide information on the information model of the **HEIDENHAIN OPC UA NC Servers**:

- **HEIDENHAIN homepage**
- **HEIDENHAIN Klartext Portal**
- **TNCguide**

Technical notes

In order to establish a connection, the OPC UA client must support the **security policy** and authentication method used by the **HEIDENHAIN OPC UA NC Server**.

The **HEIDENHAIN OPC UA NC Server** has the following end point configuration:

- **Security Mode: SignAndEncrypt**
- **Algorithm: Basic256Sha256**
- **User Authentication: X509 certificates**

ℹ The end point configuration including the URL, which depends on the server host name, is displayed on the last page of the **Connection Assistant**.

The so-called user certificate can be assigned to a user in user administration.

PKI Admin

For OPC UA, three different types of certificates are required. The server and the client need two of them, the so-called application instance certificates, in order to establish a secure connection. The third certificate is required for authorization and for starting a session with specific user rights.

There are two possibilities how to create the certificate used by the server for authentication to the client applications:

- The system generates a two-level certificate chain, the **chain of trust**

  ℹ️ This certificate chain consists in a so-called self-signed root certificate (including a **revocation list**) and a certificate for the server that was created on the basis of the root certificate.

- As an alternative, you can import a certificate and its private key that has been created specifically for the server

  ℹ️ In this case, this certificate or the certificate chain must also be provided to the client applications.

The certificate that the client uses for authentication to the server must be added on the **Trusted** tab.

All other certificates should be added on the **Issuers** tab for verification of the entire certificate chain.

ℹ️ In order to establish a connection, another user certificate is required that can be managed within the HEROS **Current User** or **UserAdmin** function.

This certificate must be added to the certificate list of an internal user with **CERTIFICATE AND KEYS** and be enabled for OPC UA. To add and enable the certificate for OPC UA, you need to import it and then select **Use for OPC-UA** on the **Certificates** tab. For a session that is initiated with this certificate, the rights of the associated internal user apply.

12.10 Changing the HEROS conversational language

The HEROS conversational language is internally based on the NC conversational language. Therefore the HEROS menu and the control cannot permanently be set to two different conversational languages.

When you change the NC conversational language, the HEROS conversational language will only reflect this change after a restart of the control.

> **ⓘ** With the optional machine parameter **applyCfgLanguage** (no. 101305), you define the behavior of the control if the NC conversational language and in the HEROS conversational language do not match.

The following link provides information on how to change the NC conversational language:

Further information: "List of user parameter", Page 552

Changing the language layout of the keyboard

You can change the language layout of the keyboard for HEROS applications.

Proceed as follows to change the language layout of the keyboard for HEROS applications:

- ▶ Select the HEROS menu icon
- ▶ Select **Settings**
- ▶ Select **Language/Keyboards**
- > The control opens the **helocale** window.
- ▶ Select the **Keyboards** tab
- ▶ Select the desired keyboard layout
- ▶ Select **Apply**
- ▶ Select **OK**
- ▶ Press **Apply**
- > The changes are applied.

13

Operating the touchscreen

13.1 Display unit and operation

Touchscreen

> ⚙ Refer to your machine manual!
> This function must be enabled and adapted by the machine tool builder.

The touchscreen is distinguished by a black frame and the lack of soft-key selection keys.

As an alternative, the TNC 640 has its operating panel integrated in the 19" screen.

1 Header

   When the control is on, the screen displays the selected operating modes in the header.

2 Soft-key row for the machine tool builder

3 Soft-key row

   The control shows further functions in a soft-key row. The active soft-key row is shown as a blue bar.

4 Integrated operating panel

5 Setting the screen layout

6 Switchover between machine operating modes, programming modes, and a third desktop

Operating panel

Depending on the version, the control can still be operated through the operating panel. Touch operation with gestures works as well.

If you have a control with integrated operating panel, the following description applies:

Integrated operating panel

The operating panel is integrated in the screen. The content of the operating panel changes depending on the current operating mode.

1   Area for showing the following:
   - Alphabetic keyboard
   - HEROS menu
   - Potentiometer for the speed of simulation (only in the **Test Run** operating mode)

2   Machine operating modes

3   Programming modes

   The control shows the active operating mode, to which the screen is switched, with a green background.

   The control shows the operating mode in the background through a small white triangle.

4
   - File management
   - Calculator
   - MOD function
   - HELP function
   - Show error messages

5   Rapid access menu

   Depending on the operating mode, you'll find the most important functions here at a glance.

6   Opening the programming dialogs (only in the **Programming** and **Positioning w/ Manual Data Input** operating modes)

7   Numerical input and axis selection

8   Navigation

9   Arrows and the jump statement **GOTO**

10   Task bar

   Further information: "Icons of the taskbar", Page 546

In addition, the machine tool builder supplies a machine operating panel.
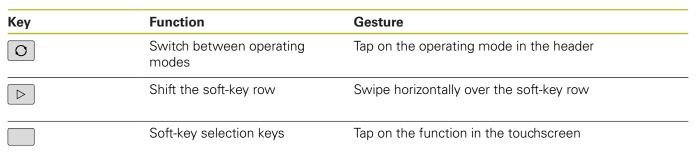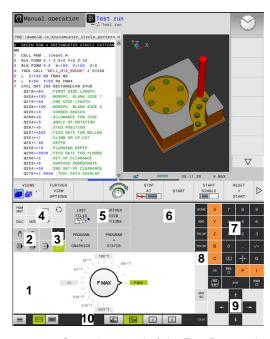
> ⚙ Refer to your machine manual!
>
> External keys, e.g. **NC START** or **NC STOP**, are described in your machine manual.

Operating panel of the Test Run mode

Operating panel in the Manual Operation mode

Basic operation

The following keys, for example, can easily be replaced by hand gestures:

Key	Function	Gesture
⟳	Switch between operating modes	Tap on the operating mode in the header
▷	Shift the soft-key row	Swipe horizontally over the soft-key row
⬜	Soft-key selection keys	Tap on the function in the touchscreen

13.2 Gestures

Overview of possible gestures

The screen of the control is multi-touch capable. That means that it can distinguish various gestures, including with two or more fingers at once.

Symbol	Gesture	Meaning
	Tap	A brief touch by a finger on the screen
	Double tap	Two brief touches on the screen
	Long press	Continuous contact of fingertip on the screen
	Swipe	Flowing motion over the screen
	Drag	A combination of long-press and then swipe, moving a finger over the screen when the starting point is clearly defined

Symbol	Gesture	Meaning
	Two-finger drag	A combination of long-press and then swipe, moving two fingers in parallel over the screen when the starting point is clearly defined
	Spread	Two fingers long-press and move away from each other
	Pinch	Two fingers move toward each other

Navigating in the table and NC programs

You can navigate in an NC program or a table as follows:

Symbol	Gesture	Function
	Tap	Mark the NC block or table line Stop scrolling
	Double tap	Activate the table line
	Swipe	Scroll through the NC program or table

Operating the simulation

The control offers touch operation with the following graphics:

- Programming graphics in the **Programming** operating mode
- 3-D view in the **Test Run** operating mode
- 3-D view in the **Program Run Single Block** operating mode
- 3-D view in the **Program Run Full Sequence** operating mode
- Kinematics view

Rotate, zoom or move a graphic

The control supports the following gestures:

Symbol	Gesture	Function
	Double tap	Set the graphic to its original size
	Drag	Rotate the graphic (only 3-D graphics)
	Two-finger drag	Move graphics
	Spread	Magnify the graphic
	Pinch	Reduce the graphic

Measure the graphic

If you have activated measurement in the **Test Run** operating mode, you have the following additional function:

Symbol	Gesture	Function
	Tap	Select the measuring point

Using the HEROS menu

You can use the **HEROS menu** menu as follows:

Symbol	Gesture	Function
	Tap	Select the application
	Long press	Open the application

Operating the CAD viewer

The control also supports touch operation for working with the
CAD-Viewer. You have various gestures available depending on the
operating mode.

To be able to use all applications, first use the icon to select the
desired function:

Icon	Function
![cursor]	Default setting
![plus]	**Add** Works in the selection mode like a pressed **Shift** key
![minus]	**Remove** Works in the selection mode like a pressed **CTRL** key

Layer setting mode and specify the workpiece preset
The control supports the following gestures:

Symbol	Gesture	Function
![circle]	Tap on an element	Show element information Specify the workpiece preset
![double-tap circle]	Double-tap on the background	Set the graphic or 3-D model to its original size
![circle] ![plus]	Activate **Add** and double-tap on the background	Reset the graphic or 3-D model to its original size and angle
![drag]	Drag	Rotate the graphic or 3-D model (only in the Layer Setting mode)

Symbol	Gesture	Function
	Two-finger drag	Move a graphic or 3-D model
	Spread	Enlarge a graphic or 3-D model
	Pinch	Reduce a graphic or 3-D model

Selecting a contour

The control supports the following gestures:

Symbol	Gesture	Function
	Tap on an element	Select element
	Tap on an element in the list-view window	Select or deselect an element
	Activate **Add** and tap on an element	Part, shorten, or lengthen and element
	Activate **Remove** and tap on an element	Deselect an element
	Double-tap on the background	Reset the graphic to its original size
	Swipe over an element	Show a preview of selected elements Show element information

Symbol	Gesture	Function
	Two-finger drag	Move graphics
	Spread	Magnify the graphic
	Pinch	Reduce the graphic

Selecting machining positions

The control supports the following gestures:

Symbol	Gesture	Function
	Tap on an element	Select element Selecting an intersection
	Double-tap on the background	Reset the graphic to its original size
	Swipe over an element	Show a preview of selected elements Show element information
	Activate **Add** and drag	Spread a fast selection area
	Activate **Remove** and drag	Spread an area for deselection of elements
	Two-finger drag	Move graphics

Symbol	Gesture	Function
	Spread	Magnify the graphic
	Pinch	Reduce the graphic

Save elements and switch to the NC program

When you tap on the appropriate icons, the controls saves the selected elements.

You can switch back to the **Programming** operating mode in the following ways:

- Press the **Programming** key

  The control switches to the **Programming** mode of operation.

- Close the **CAD-Viewer**

  The control automatically switches to the **Programming** operating mode.

- Use the task bar to leave the **CAD-Viewer** open on the third desktop

  The third desktop stays active in the background

13.3 Functions in the taskbar

Icons of the taskbar

The taskbar provides you with the following icons:

Icon	Function
	Open the HEROS menu
AUTO	Automatically show and hide the alphabetic keyboard
	Always show the alphabetic keyboard
	Workspace 1: Select the active machine mode
	Workspace 2: Select the active programming mode
3	Workspace 3: Select the CAD viewer, DXF converter or applications of the machine tool builder (optionally available)
4	Workspace 4: Select the display and remote control of external computer units (Option 133) or applications of the machine tool builder (optionally available)
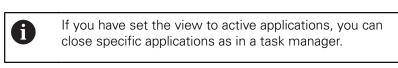
Functions in the HEROS menu

Using the **Menu** icon on the taskbar, you can open the HEROS menu through which you can find information, make settings or start applications.

Further information: "Overview of taskbar", Page 459

The taskbar provides you with the following icons:

Icon	Function
←	Return to main menu
	Show active applications
	Show all applications

i	If you have set the view to active applications, you can close specific applications as in a task manager.

Touchscreen calibration

With the **Touchscreen Calibration** function you can measure and correct the screen.

Calibrating the touchscreen

Proceed as follows to conduct the functions:

▶ Use the **DIADUR** icon to open the HEROS menu

▶ Select the **Touchscreen Calibration** menu item

> The control starts the calibration mode.

▶ Successively tap on the blinking symbols

If you would like to stop the calibration:

▶ Press the **ESC** key

Touchscreen configuration

With the **Touchscreen Configuration** function you can define the properties of the screen.

Adjusting sensitivity

Proceed as follows to adjust the sensitivity:

▶ Use the **DIADUR** icon to open the HeROS menu

▶ Select the **Touchscreen Configuration** menu item

> The control opens a pop-up window.

▶ Select the sensitivity

▶ Confirm with **OK**

Display of the touch points

Proceed as follows to show or hide the touch points:

▶ Press the **DIADUR** key to open the JH menu

▶ Select the **Touchscreen Configuration** menu item

> The control opens a pop-up window.

▶ Select the **Show Touch Points** display

   ■ **Disable Touchfingers** to hide the touch points

   ■ **Enable Single Touchfinger** to show the touch point

   ■ **Enable Full Touchfingers** to show the touch points of all fingers involved

▶ Confirm with **OK**

Touchscreen cleaning

With the **Touchscreen Cleaning** function you can lock the screen in order to clean it.

Activating the cleaning mode

Proceed as follows to change activate the cleaning mode:

▶   Use the **DIADUR** icon to open the HEROS menu

▶   Select the **Touchscreen Cleaning** menu item

>   The control locks the screen for 90 seconds.

▶   Clean the screen

If you would like to stop the cleaning mode:

▶   Pull the displayed sliders apart at the same time

14

Tables and overviews

14.1 Machine-specific user parameters

Application

The parameter values are entered in the **configuration editor**.

Refer to your machine manual!

■ The machine tool builder can make additional machine-specific parameters available as user parameters, so that you can configure the functions that are available.

■ The machine tool builder can adapt the structure and contents of the user parameters. The display on your machine may be different.

The machine parameters are grouped as parameter objects in a tree structure in the configuration editor. Each parameter object has a name (e.g. **Settings for screen displays**) that gives information about the parameters it contains.

Calling the configuration editor

Proceed as follows:

| MOD | ▶ Press the **MOD** key

| ↑ | ▶ If required, use the arrow keys to navigate to **Code-number entry**

▶ Enter the code number **123**

| ENT | ▶ Confirm with the **ENT** key

> The control displays the list of available parameters in the tree view.

Depiction of the parameters

The icon at the beginning of each line in the parameter tree shows additional information about this line. The icons have the following meanings:

- ▪ 　 **Branch exists but is closed**

- ▪ 　 **Branch is open**

- ▪ 　 **Empty object, cannot be opened**

- ▪ 　 **Initialized machine parameter**

- ▪ 　 **Uninitialized (optional) machine parameter**

- ▪ 　 **Can be read but not edited**

- ▪ 　 **Can neither be read nor edited**

The folder icon indicates the object type:

- ▪ 　 **Key (group name)**

- ▪ 　 **List**

- ▪ 　 **Entity (parameter object)**

> ℹ️ The icons of not yet active parameters and objects appear dimmed. These can be activated with the **MORE FUNCTIONS** and **INSERT** soft keys.

Changing the parameters

Proceed as follows:

- ▶ Find the desired parameter
- ▶ Edit the value

END

- ▶ Press the **END** soft key to close the configuration editor

STORE

- ▶ Press **SAVE** soft key to save the changes

> ℹ️ The control saves a change list of the last 20 changes to the configuration data. To undo changes, select the corresponding line and press the **MORE FUNCTIONS** and **CANCEL CHANGE** soft keys.

Changing the display of the parameters

If you are in the configuration editor for the user parameters, you can change the display of the existing parameters. In the default setting, the parameters are displayed with short, explanatory texts.

Proceed as follows in order to have the actual system names of the parameters be shown:

⊙ ▶ Press the **Screen layout** key

```
DISPLAY
THE SYS-
TEM NAME
```
▶ Press the **SHOW SYSTEM NAME** soft key

Follow the same procedure to return to the standard display.

Displaying help texts

The **HELP** key enables you to call a help text for each parameter object or attribute.

If the help text does not fit on one page (1/2 is then displayed at the upper right, for example), press the **HELP PAGE** soft key to scroll to the second page.

The control displays other information (e.g. unit of measurement, initial value, selection list) in addition to the help text. If the selected machine parameter matches a parameter in the previous control model, the corresponding MP number is displayed.

List of user parameter

⚙ Refer to your machine manual!
- The machine tool builder can make additional machine-specific parameters available as user parameters, so that you can configure the functions that are available.
- The machine tool builder can adapt the structure and contents of the user parameters. The display on your machine may be different.

Parameter settings

DisplaySettings

    Display sequence and rules for axes

        [0] to [23]

            Depends on the available axes

                Key name of an object in CfgAxis

                    Key name of the axis to be displayed

                Axis designation

                    Axis designation to be used in place of the key name

                Display rule for the axis

                    ShowAlways

                    IfKinem

                    IfKinemaxis

                    IfNotKinemAxis

                    Never

    Sequence of the axes shown in the REF display

        [0] to [23]

            Depends on the available axes

    Type of position display in the position window

        NOML

        ACTL

        REF ACTL

        REF NOML

        LAG

        ACTDST

        REFDST

        M 118

    Type of position display in the status display

        NOML

        ACTL

        REF ACTL

        REF NOML

        LAG

        ACTDST

        REFDST

        M 118

    Definition of decimal separator for position display

        . point

        , comma

Parameter settings

Feed rate display in Manual Operation mode

at axis key: Display the feed rate only if axis-direction key is pressed

always minimum: Always display the feed rate

Display of spindle position in the position display

during closed loop: Display spindle position only if spindle is in position control loop

during closed loop and M5: Display spindle position if spindle is in position control loop and with M5

Lock the PRESET MANAGEMENT soft key

True: Access to the preset table is locked

False: The preset table can be accessed by soft key

Font size for program display

FONT_APPLICATION_SMALL

FONT_APPLICATION_MEDIUM

Sequence of icons in the display

[0] to [19]

Depends on the activated options

Settings for display behavior

Depends on the machine tool builder

DisplaySettings

Display step for the individual axes

List of all available axes

Display step for position display in mm or degrees

0.1

0.05

0.01

0.005

0.001

0.0005

0.0001

0.00005

0.00001

Display step for position display in inches

0.005

0.001

0.0005

0.0001

0.00005

0.00001

Parameter settings

DisplaySettings

Definition of the unit of measure valid for the display

metric: Use the metric system

inch: Use the inch system

DisplaySettings

Format of the NC programs and cycle display

Program entry in HEIDENHAIN Klartext conversational programming language or in DIN/ISO

HEIDENHAIN: Klartext conversational programming language for entering the program in Positioning with Manual Data Input operating mode

ISO: DIN/ISO for entering the program in Positioning with Manual Data Input operating mode

---

Parameter settings

DisplaySettings

    Definition of the NC and PLC conversational languages

        NC conversational language

            ENGLISH

            GERMAN

            CZECH

            FRENCH

            ITALIAN

            SPANISH

            PORTUGUESE

            SWEDISH

            DANISH

            FINNISH

            DUTCH

            POLISH

            HUNGARIAN

            RUSSIAN

            CHINESE

            CHINESE_TRAD

            SLOVENIAN

            KOREAN

            NORWEGIAN

            ROMANIAN

            SLOVAK

            TURKISH

        Load the language of the NC control

            FALSE: When starting the control, the language of the HEROS operating system is used

            TRUE: When starting the control, the language from the machine parameters is used

        PLC conversational language

            See NC conversational language

        PLC error message language

            See NC conversational language

        Language for online help

            See NC conversational language

DisplaySettings

    Behavior during control start-up

        Acknowledge the "Power interrupted" message

            TRUE: Start-up of the control is not continued until the message has been acknowledged

            FALSE: The Power interrupted message does not appear

---

Parameter settings

DisplaySettings
    Display mode

        Analog
        Digital
        Logo
        Analog and logo
        Digital and logo
        Analog on logo
        Digital on logo

DisplaySettings
    Link row on/off
        Display settings for link row
            OFF: Switch off the information line in the operating-mode line
            ON: Switch on the information line in the operating-mode line

DisplaySettings
    Settings for 3-D view
        Model type of the 3-D view
            3-D (compute-intensive): Model depiction for complex machining operations with undercuts
            2.5-D: Model depiction for 3-axis operations
            No Model: Model depiction is deactivated

        Model quality of 3-D view
            very high: High resolution; the block end points can be depicted
            high: High resolution
            medium: Medium resolution
            low: Low resolution

        Reset tool paths in new BLK form
            ON: With new BLK form in Test Run, the tool paths are reset
            OFF: With new BLK form in Test Run, the tool paths are not reset

        Generate graphics journal data after restart
            OFF: Do not generate journal data
            ON: Generate journal data after a restart for diagnostic purposes

Parameter settings

DisplaySettings

    Position display settings

        Position display

        With TOOL CALL DL

            As Tool Length: The programmed oversize DL is considered to be the change in tool length for display of the workpiece-based position

            As Workpiece Oversize: The programmed oversize DL is considered to be the workpiece oversize for display of the workpiece-based position

DisplaySettings

Table editor setting

    Behavior when deleting tools from the pocket table

        DISABLED: Deletion of tool is not possible

        WITH_WARNING: Tool deletion is possible, but must be confirmed

        WITHOUT_WARNING: Tool deletion is possible without warning

    Behavior when deleting a tool's index entries

        ALWAYS_ALLOWED: Deletion of index entries is always possible

        TOOL_RULES: The behavior depends on the setting of the parameter "Behavior when deleting tools from the pocket table"

    Show the RESET COLUMN T soft key

        TRUE: The soft key is displayed and all tools can be deleted from the tool memory by the user

        FALSE: The soft key is not displayed

DisplaySettings

Setting the coordinate systems for the display

    Coordinate system for the datum shift

        WorkplaneSystem: Datum is displayed in the system of the tilted plane, WPL-CS

        WorkpieceSystem: Datum is displayed in the workpiece coordinate system, W-CS

Parameter settings

DisplaySettings

GPS display settings

Show offset in the GPS dialog

OFF: The offsets are not shown in the GPS dialog

ON : The offsets are shown in the GPS dialog

Show an additive basic rotation in the GPS dialog

OFF: Do not show the additive basic rotation in the GPS dialog

ON: Show the additive basic rotation in the GPS dialog

Show shift of W-CS in the GPS dialog

OFF: Do not show the shift of W-CS in the GPS dialog

ON: Show the shift of W-CS in the GPS dialog

Show mirror image in the GPS dialog

OFF: Do not show the mirror image in the GPS dialog

ON: Show the mirror image in the GPS dialog

Show shift of mW-CS in the GPS dialog

OFF: Do not show the shift of mW-CS in the GPS dialog

ON: Show the shift of mW-CS in the GPS dialog

Show rotation in the GPS dialog

OFF: Do not show the rotation in the GPS dialog

ON: Show the rotation in the GPS dialog

Show feed rate the GPS dialog

OFF: Do not show the feed rate in the GPS dialog

ON: Show the feed rate in the GPS dialog

M-CS coordinate system is selectable

OFF: M-CS coordinate system is not selectable

ON: M-CS coordinate system is selectable

W-CS coordinate system is selectable

OFF: W-CS coordinate system is not selectable

ON: W-CS coordinate system is selectable

mM-CS coordinate system is selectable

OFF: mM-CS coordinate system is not selectable

ON: mM-CS coordinate system is selectable

WPL-CS coordinate system is selectable

OFF: WPL-CS coordinate system is not selectable

ON: WPL-CS coordinate system is selectable

Parameter settings

ProbeSettings

    Configuration of the tool calibration

        TT140_1

            M function for spindle orientation

                –1: Spindle orientation directly by NC

                0: Function inactive

                1 to 999: Number of the M function for spindle orientation

            Probing routine

                MultiDirections: Probing from multiple directions

                SingleDirection: Probing from one direction

            Probing direction for tool radius measurement

                X_Positive, Y_Positive, X_Negative, Y_Negative, Z_Positive, Z_Negative (depending on the tool axis)

            Distance from lower edge of tool to upper edge of stylus

                0.001 to 99.9999 [mm]: Offset of stylus to tool

            Rapid traverse in probing cycle

                10 to 300 000 [mm/min]: Rapid traverse in probing cycle

            Probing feed rate during tool measurement

                1 to 3000 [mm/min]: Probing feed rate during tool measurement

            Calculation of the probing feed rate

                ConstantTolerance: Calculation of the probing feed rate with constant tolerance

                VariableTolerance: Calculation of the probing feed rate with variable tolerance

                ConstantFeed: Constant probing feed rate

            Speed determination method

                Automatic: Automatically determine speed

                MinSpindleSpeed: Use the minimum spindle speed

            Max. permissible surface cutting speed at the tooth edge

                1 to129 [m/min]: Permissible surface cutting speed at the circumference of the milling tool

            Maximum permissible speed for tool measurement

                0 to 1000 [1/min]: Maximum permissible speed

            Maximum permissible measurement error during tool measurement

                0.001 to 0.999 [mm]: First maximum permissible measurement error

            Maximum permissible measurement error during tool measurement

                0.001 to 0.999 [mm]: Second maximum permissible measurement error

Parameter settings

NC stop during tool check

True: When the breakage tolerance is exceeded, the NC program is stopped

False: The NC program is not stopped

NC stop during tool measurement

True: When the breakage tolerance is exceeded, the NC program is stopped

False: The NC program is not stopped

Change the tool table during tool check and tool measurement

AdaptOnMeasure: The table is changed after tool measurement

AdaptOnBoth: The table is changed after tool check and tool measurement

AdaptNever: The table is not changed after tool check and tool measurement

Configuration of a round stylus

TT140_1

Coordinates of the stylus center

[0]: X coordinate of the stylus center with respect to the machine datum

[1]: Y coordinate of the stylus center with respect to the machine datum

[2]: Z coordinate of the stylus center with respect to the machine datum

Set-up clearance above the stylus for pre-positioning

0.001 to 99 999.9999 [mm] set-up clearance

Safety zone around the stylus for pre-positioning

0.001 to 99 999.9999 [mm]: Set-up clearance in the plane perpendicular to the tool axis

Parameter settings

ChannelSettings

CH_NC

Active kinematics

Kinematics to be activated

List of machine kinematics

The kinematics to be activated during control start-up

List of machine kinematics

Specify the behavior of the NC program

Reset the machining time when program starts

True: Machining time is reset

False: Machining time is not reset

PLC signal for the number of the pending fixed cycle

Depends on the machine tool builder

Geometry tolerances

Permissible deviation of the circle radius

0.0001 to 0.016 [mm]: Permissible deviation of the radius at the circle end-point compared with the circle start-point

Permissible error in successive threads

0.0001 to 999.9999 [mm]: Permissible deviation of the dynamically smoothed contour from the programmed thread contour

Reserve for retraction movements

0.0001 to 10 [mm]: Distance to limit switch or collision object with M140 MB MAX

Configuration of the fixed cycles

Overlap factor for pocket milling

0.001 to 1.414: Overlap factor for Cycle 4 POCKET MILLING and Cycle 5 CIRCU-LAR POCKET

Traverse after machining a contour pocket

PosBeforeMachining: Tool moves to the same position as before the cycle was machined

ToolAxClearanceHeight: Position the tool axis to clearance height

Display the **Spindle ?** error message if M3/M4 is not active

On: Issue error message

Off: No error message

Display the **Enter depth as negative**

On: Issue error message

Off: No error message

Parameter settings

Behavior when moving to wall of slot in the cylinder surface
>   **LineNormal: Approach on a straight line**
>   **CircleTangential: Approach**

M function for spindle orientation
>   **–1: Spindle orientation directly by NC**
>   **0: Function inactive**
>   **1 to 999: Number of the M function for spindle orientation**

Do not display **Plunging type is not possible** error message
>   **On: Error message is not displayed**
>   **Off: Error message is displayed**

Behavior of M7 and M8 with Cycles 202 and 204
>   **TRUE: At the end of Cycles 202 and 204, the status of M7 and M8 is restored to that before the cycle call**
>   **FALSE: At the end of Cycles 202 and 204, the status of M7 and M8 is not restored automatically**

Automatic feed rate reduction after attaining SMAX
>   **100: Feed rate reduction is deactivated**
>   **0 < factor < 100: Feed rate reduction is activated. Minimum feed rate in percent of the programmed feed rate in the turning cycle**

Do not show the **Residual material** warning
>   **On: Warning is not displayed**
>   **Off: Warning is displayed**

Geometry filter for culling linear elements
> Type of stretch filter
>>   **- Off: No filter active**
>>   **- ShortCut: Omit individual points on a polygon**
>>   **- Average: The geometry filter smooths corners**

> Maximum distance of the filtered to the unfiltered contour
>>   **0 to 10 [mm]: The filtered points lie within this tolerance to the resulting new path**

> Maximum length of the distance resulting from filtering
>>   **0 to 1000 [mm]: Length over which geometry filtering is active**

Special spindle parameters
> Potentiometer for feed rate during thread cutting
>>   **SpindlePotentiometer: During thread cutting, the potentiometer is effective for the spindle-speed override. The potentiometer for the feed-rate override is disabled**

Parameter settings

FeedPotentiometer: During thread cutting, the potentiometer is effective for the feed-rate override. The potentiometer for the spindle-speed override is not active

Waiting time at reversal point in thread base

–999999999 to 999999999: The spindle stops for this time at the bottom of the thread before starting again in the opposite direction of rotation

Advanced switching time of spindle

–999999999 to 999999999: The spindle is stopped at this time before reaching the bottom of the thread

Limitation of spindle speed with Cycles 17, 207, and 18

TRUE: With small thread depths, the spindle speed is limited so that it runs at constant speed approx. 1/3 of the time
FALSE: No limitation of the spindle speed

Parameter settings

Settings for the NC editor

Generate backup files

TRUE: Generate backup file after editing NC programs

FALSE: Do not generate backup file after editing NC programs

Behavior of the cursor after deletion of lines

TRUE: Cursor is placed on the preceding line after deletion (behavior of iTNC)

FALSE: Cursor is placed on the following line after deletion

Behavior of the cursor on the first or last line

TRUE: Cursor jumps from end to beginning of program

FALSE: Cursor is not allowed to jump from end to beginning of program

Line break on blocks with more than one line

ALL: Always display all lines

ACT: Only display the lines of the active block completely

NO: Only display all lines when block is edited

Activate help graphics when entering cycle data

TRUE: Always display help graphics during input

FALSE: Only display help graphics when the CYCLE HELP soft key is set to ON. The CYCLE HELP OFF/ON soft key is displayed in Programming operating mode after pressing the Screen layout key

Behavior of the soft-key row after entering a cycle

TRUE: The cycle soft-key row remains active after a cycle definition

FALSE: The cycle soft-key row is hidden after a cycle definition

Confirmation request when deleting a block

TRUE: Display confirmation request when deleting an NC block

FALSE: Do not display confirmation request when deleting an NC block

Line number up to which a test of the NC program is to be run.

100 to 100000: Program length for which the geometry is to be checked

ISO programming: Block number increment

0 to 250: Increments for creation of ISO blocks in the program

Specify programmable axes

TRUE: Use the specified axis configuration

FALSE: Use the default axis configuration XYZABCUVW

Behavior of paraxial positioning blocks

TRUE: Paraxial positioning blocks enabled

FALSE: Paraxial positioning blocks disabled

Line number to which identical syntax elements are searched for

500 to 400000: Search for selected elements with up or down arrow keys

Parameter settings

Behavior of PARAXMODE function with UVW axes

FALSE: PARAXMODE function enabled

TRUE: PARAXMODE function disabled

File management settings

Display of dependent files

MANUAL: Dependent files are displayed

AUTOMATIC: Dependent files are not displayed

Settings for tool-usage files

Create tool-usage file for NC program

NotAutoCreate: Do not create a tool usage list when a program is selected

OnProgSelectionIfNotExist: When a program is selected, create a list if does not yet exist

OnProgSelectionIfNecessary: When a program is selected, create a list if it does not yet exist or is obsolete

OnProgSelectionAndModify: When a program is selected, create a list if it does not yet exist or the program is changed

Create a pallet usage file

NotAutoCreate: Do not create a tool usage list when a pallet is selected

OnProgSelectionIfNotExist: When a pallet is selected, a list is created if it does not yet exist

OnProgSelectionIfNecessary: When a pallet is selected, a list is created if it does not yet exist or is obsolete

OnProgSelectionAndModify: When a pallet is selected, a list is created if it does not yet exist, is obsolete or the program is changed

Paths for the end user

These machine parameters are only effective for a Windows programming station

List of drives and/or directories

Drives or directories entered here are shown in the control's file manager

FN 16 output path for execution

Path for FN 16 output when no path is defined in the NC program

FN 16 output path for the Programming and Test Run operating modes

Path for FN 16 output when no path is defined in the NC program

Serial Interface RS232

Further information: "Setting up data interfaces", Page 479

Parameter settings

Monitoring

    User settings for the component monitor

        Perform configured error reaction

            TRUE: Error reaction is performed

            FALSE: Error reaction is not performed

        Display warning for the component monitor

            TRUE: Warnings are displayed

            FALSE: Warnings are not displayed

General information on the machine operator

    Information retrievable via an interface

        Custom name (nickname) of the machine

        Inventory number or ID

        Photo or image of the machine

            Path to the image file

        Machine location

        Department or division

        Machine responsibility

        Contact email address

        Contact phone number

14.2 Pin layout and cables for data interfaces

RS-232-C/V.24 interface for HEIDENHAIN devices

ℹ The interface complies with the requirements of EN 50178 for **Low voltage electrical separation**.

When using the 25-pin adapter block:

Control		Conn. cable 365725-xx			Adapter block 310085-01		Conn. cable 274545-xx		
Male	Assignment	Female	Color	Female	Male	Female	Male	Color	Female
1	Do not assign	1		1	1	1	1	White/Brown	1
2	RXD	2	Yellow	3	3	3	3	Yellow	2
3	TXD	3	Green	2	2	2	2	Green	3
4	DTR	4	Brown	20	20	20	20	Brown	8
5	Signal GND	5	Red	7	7	7	7	Red	7
6	DSR	6	Blue	6	6	6	6		6
7	RTS	7	Gray	4	4	4	4	Gray	5
8	CTR	8	Pink	5	5	5	5	Pink	4
9	Do not assign	9					8	Violet	20
Hsg.	External shield	Hsg.	External shield	Hsg.	Hsg.	Hsg.	Hsg.	External shield	Hsg.

When using the 9-pin adapter block:

Control			Conn. cable 355484-xx			Adapter block 363987-02			Conn. cable 366964-xx		
Male	**Assign-ment**	**Female**	**Color**	**Male**	**Female**	**Male**	**Female**	**Color**	**Female**		
1	Do not assign	1	Red	1	1	1	1	Red	1		
2	RXD	2	Yellow	2	2	2	2	Yellow	3		
3	TXD	3	White	3	3	3	3	White	2		
4	DTR	4	Brown	4	4	4	4	Brown	6		
5	Signal GND	5	Black	5	5	5	5	Black	5		
6	DSR	6	Violet	6	6	6	6	Violet	4		
7	RTS	7	Gray	7	7	7	7	Gray	8		
8	CTR	8	White/Green	8	8	8	8	White/Green	7		
9	Do not assign	9	Green	9	9	9	9	Green	9		
Hsg.	External shield	Hsg.	External shield	Hsg.	Hsg.	Hsg.	Hsg.	External shield	Hsg.		

Non-HEIDENHAIN devices

The connector layout of a non-HEIDENHAIN device may substantially differ from that of a HEIDENHAIN device.

It depends on the unit and the type of data transfer. The table below shows the connector pin layout on the adapter block.

Adapter block 363987-02		Conn. cable 366964-xx		
Female	**Male**	**Female**	**Color**	**Female**
1	1	1	Red	1
2	2	2	Yellow	3
3	3	3	White	2
4	4	4	Brown	6
5	5	5	Black	5
6	6	6	Violet	4
7	7	7	Gray	8
8	8	8	White/Green	7
9	9	9	Green	9
Hsg.	Hsg.	Hsg.	External shield	Hsg.

Ethernet interface RJ45 socket

Maximum cable length:

- Unshielded: 100 m
- Shielded: 400 m

Pin	Signal	Description
1	TX+	Transmit Data
2	TX–	Transmit Data
3	REC+	Receive Data
4	Vacant	
5	Vacant	
6	REC–	Receive Data
7	Vacant	
8	Vacant	

14.3 Specifications

Explanation of symbols

> ⓘ **Further information:**
> TNC 640 brochure, ID: 892916-xx brochure
> TNC 640 HSCI brochure, ID: 896020-xx

- ■ Default
- □ Axis option
- **1** Advanced Function Set 1
- **2** Advanced Function Set 2

Specifications

Specifications	
Components	■ Operating panel
	■ Screen with soft keys or touchscreen
Program memory	■ Minimum 21 GB
Input resolution and display step	■ As fine as 0.1 µm for linear axes
	■ As fine as 0.01 µm for linear axes
	■ Up to 0.0001° for rotary axes
	■ As fine as 0.000 01° for rotary axes
Input range	■ Maximum 999 999 999 mm or 999 999 999°
Interpolation	■ Linear in 4 axes
	■ Circular in 2 axes
	■ Helical: superimposition of circular and straight paths
Block processing time 3-D straight line without radius compensation	■ 0.5 ms
Axis feedback control	■ Position loop resolution: Signal period of the position encoder/1024
	■ Cycle time of position controller: 3 ms
	■ Cycle time of speed controller: 200 µs
Range of traverse	■ Maximum 100 m (3937 inches)
Spindle speed	■ Maximum 100,000 rpm (analog speed command signal)
Error compensation	■ Linear and nonlinear axis error, backlash, reversal peaks during circular movements, thermal expansion
	■ Static friction

Specifications

Data interfaces	■ One each RS-232-C /V.24 max. 115 kilobaud
	■ Extended data interface with LSV2 protocol for remote operation of the control through the data interface with the **TNCremo** software
	■ Ethernet interface 1000 BaseT
	■ 5 x USB (1 x front USB 2.0; 4 x rear USB 3.0)
Ambient temperature	■ Operation: 5 °C to +40 °C
	■ Storage: -20 °C to +60 °C

Input formats and units of control functions

Positions, coordinates, circle radii, chamfer lengths	–99 999.9999 to +99 999.9999 (5,4: number of digits before and after the decimal point) [mm]
Tool numbers	0 to 32 767.9 (5, 1)
Tool names	32 characters, enclosed by quotation marks with **TOOL CALL**. Permitted special characters: # $ % & . , - _
Detail values for tool compensation	–99.9999 to +99.9999 (2, 4) [mm]
Spindle speeds	0 to 99 999.999 (5, 3) [rpm]
Feed rates	0 to 99,999.999 (5, 3) [mm/min] or [mm/tooth] or [mm/1]
Dwell time in Cycle 9	0 to 3600.000 (4, 3) [s]
Thread pitch in various cycles	-9.9999 to +9.9999 (2, 4) [mm]
Angle for spindle orientation	0 to 360.0000 (3, 4) [°]
Angle for polar coordinates, rotation, tilting the working plane	-360.0000 to 360.0000 (3, 4) [°]
Polar coordination for screw line interpolation (CP)	–5 400.0000 to 5 400.0000 (4, 4) [°]
Datum numbers in Cycle 7	0 to 2999 (4, 0)
Scaling factor in Cycles 11 and 26	0.000001 to 99.999999 (2, 6)
Miscellaneous functions M	0 to 9999 (4.0)
Q parameter numbers	0 to 1999 (4, 0)
Q parameter values	–99 999.9999 to +99 999.9999 (9, 6)
Surface-normal vectors N and T with 3-D compensation	–9.99999999 to +9.99999999 (1, 8)
Labels (LBL) for program jumps	0 to 65535 (5.0)
Labels (LBL) for program jumps	Any text string in quotation marks ("")
Number of program section repeats REP	1 to 65 534 (5, 0)
Error number for Q parameter function FN 14	0 to 1199 (4, 0)

User functions

User functions		
Short description	■	Basic version: 3 axes plus closed-loop spindle
	■	Fourth NC axis plus auxiliary axis
		or
	□	8 additional axes or 7 additional axes plus 2nd spindle
	■	Digital current and speed control
Program entry	In HEIDENHAIN conversational format and DIN/ISO	
Position entry	■	Nominal positions for lines and arcs in Cartesian coordinates or polar coordinates
	■	Incremental or absolute dimensions
	■	Display and entry in mm or inches
Tool compensation	■	Tool radius in the working plane and tool length
	■	Radius-compensated contour look-ahead for up to 99 NC blocks (M120)
	2	Three-dimensional tool-radius compensation for changing tool data without having to recalculate an existing NC program
Tool tables	Multiple tool tables with any number of tools	
Constant contour speed	■	With respect to the path of the tool center
	■	With respect to the cutting edge
Parallel operation	Creating an NC program with graphical support while another NC program is being run	
3-D machining (Advanced Function Set 2)	**2**	Motion control with minimum jerk
	2	3-D tool compensation through surface-normal vectors
	2	Using the electronic handwheel to change the angle of the swivel head during program run without affecting the position of the tool center point (tool tip or center of sphere) (TCPM = **T**ool **C**enter **P**oint **M**anagement)
	2	Keeping the tool normal to the contour
	2	Tool radius compensation perpendicular to traversing and tool direction
Rotary table machining (Advanced Function Set 1)	**1**	Programming of cylindrical contours as if in two axes
	1	Feed rate in distance per minute

User functions		
Contour elements	■	Straight line
	■	Chamfer
	■	Circular path
	■	Circle center
	■	Circle radius
	■	Tangentially connected arc
	■	Rounded corners
Approaching and departing the contour	■	Via straight line: tangential or perpendicular
	■	Via circular arc
FK free contour programming	■	FK free contour programming in HEIDENHAIN conversational format with graphic support for workpiece drawings not dimensioned for NC
Program jumps	■	Subprograms
	■	Program section repeats
	■	External NC programs
Machining cycles	■	Cycles for drilling, and conventional and rigid tapping
	■	Roughing of rectangular and circular pockets
	■	Cycles for pecking, reaming, boring, and counterboring
	■	Cycles for milling internal and external threads
	■	Finishing of rectangular and circular pockets
	■	Cycles for clearing level and inclined surfaces
	■	Cycles for milling linear and circular slots
	■	Cartesian and polar point patterns and point patterns for DataMatrix code
	■	Contour-parallel contour pocket
	■	Contour train
	■	Cycles for turning operations
	■	OEM cycles (special cycles developed by the machine manufacturer) can also be integrated
Coordinate transformation	■	Datum shift, rotation, mirroring
	■	Scaling factor (axis-specific)
	1	Tilting the working plane (Advanced Function Set 1)

User functions

Q parameters Programming with variables	■ Mathematical functions: =, +, –, *, sin α, cos α, root
	■ Logical operations (=, ≠, <, >)
	■ Calculating with parentheses
	■ tan α, arc sin, arc cos, arc tan, a^n, e^n, ln, log, absolute value of a number, constant π, negation, truncation of digits before or after the decimal point
	■ Functions for calculation of circles
	■ String parameters
Programming aids	■ Calculator
	■ Color highlighting of syntax elements
	■ Complete list of all current error messages
	■ Context-sensitive help function for error messages
	■ Graphic support for the programming of cycles
	■ Comment blocks in NC program
Teach-In	■ Actual positions can be transferred directly to the NC program
Test graphics Display modes	■ Graphic simulation before a program run, even while another NC program is being run
	■ Plan view / projection in 3 planes / 3-D view / 3-D line graphic
	■ Detail enlargement
Programming graphics	■ In the Programming mode, the contours of the NC blocks are drawn on screen while they are being entered (2-D pencil-trace graphics), even while another NC program is being run
Program-run graphics Display modes	■ Graphic simulation of real-time machining in plan view / projection in 3 planes / 3-D view
Machining time	■ Calculation of machining time in the **Test Run** operating mode
	■ Display of the current machining time in the Program Run operating modes
Contour, returning to	■ Block scan in any NC block in the NC program, returning the tool to the calculated nominal position to continue machining
	■ NC program interruption, contour departure and return
Datum tables	■ Multiple datum tables for storing workpiece-specific datums
Touch probe cycles	■ Calibrating the touch probe
	■ Compensation of workpiece misalignment, manual or automatic
	■ Presetting, manual or automatic
	■ Automatically measuring workpieces
	■ Cycles for automatic tool measurement
	■ Cycles for automatic kinematics measurement

Accessories

Accessories		
Electronic handwheels	■	HR 510: Portable handwheel
	■	HR 550 FS: Portable radio handwheel with display
	■	HR 520: Portable handwheel with display
	■	HR 130: Panel-mounted handwheel
	■	HR 150: Up to three panel-mounted handwheels via handwheel adapter HRA 110
Touch probes	■	TS 248: workpiece touch trigger probe with cable connection
	■	TS 260: workpiece touch trigger probe with cable connection
	■	TS 460: workpiece touch trigger probe with infrared or radio transmission
	■	TS 642: workpiece touch trigger probe with infrared transmission
	■	TS 740: high-precision workpiece touch trigger probe with infrared transmission
	■	TT 160: tool touch trigger probe
	■	TT 460: tool touch trigger probe with infrared transmission

14.4 Differences between the TNC 640 and the iTNC 530

> ℹ️ **Further information:**
> User's Manuals for Cycle Programming, Conversational Programming and ISO-Programming as well as Functions of the TNC 640—Comparison to the iTNC 530 brochure
> ID: 1110731-xx

Comparison: Specifications

Function	TNC 640	iTNC 530
Control loops	Maximum 24 control loops (including up to 4 spindles)	18 maximum
Input resolution and display step:		
▪ Linear axes	▪ 0.01 µm	▪ 0.1 µm
▪ Rotary axes	▪ 0.00001°	▪ 0.0001°
Display	19-inch screen with soft keys or 19-inch screen with touchscreen	19-inch screen or 15.1-inch screen with soft keys
Interpolation:		
▪ Straight line	▪ 6 axes	▪ 5 axes
▪ Circle	▪ 3 axes	▪ 3 axes
▪ Helix	▪ yes	▪ Yes
▪ Spline	▪ No	▪ Yes with option 9

Comparison: Data interfaces

Function	TNC 640	iTNC 530
RS-422/V.11 serial interface	-	X

Further information: "Setting up data interfaces", Page 479

Comparison: PC software

Function	TNC 640	iTNC 530
M3D Converter for the creation of high-resolution collision objects for collision monitoring (DCM)	Available	Not available
ConfigDesign for the configuration of machine parameters	Available	Not available
TNCanalyzer for the analysis and evaluation of service files	Available	Not available

Comparison: User functions

Function	TNC 640	iTNC 530
Program entry		
■ smarT.NC	■ –	■ X
■ ASCII editor	■ X, directly editable	■ X, editable after conversion
Position entry		
■ Set the last tool position as pole (empty CC block)	■ X (error message if pole transfer is ambiguous)	■ X
■ Spline sets (**SPL**)	■ –	■ X, with option 9
Tool table		
■ Flexible management of tool types	■ X	■ –
■ Filtered display of selectable tools	■ X	■ –
■ Sorting function	■ X	■ –
■ Column names	■ Sometimes with _	■ Sometimes with -
■ Form view	■ Switchover with Screen Layout key	■ Switchover by soft key
■ Exchange of tool table between TNC 640 and iTNC 530	■ X	■ Not possible
Touch probe table for managing different 3-D touch probes	X	–
Cutting data calculator: Automatic calculation of spindle speed and feed rate	■ Simple cutting data calculator without stored table ■ Cutting data calculator with stored technology tables	Using stored technology tables

Function	TNC 640	iTNC 530
Define any tables	■ Freely definable tables (.TAB files) ■ Reading and writing with FN functions ■ Definable via config. data ■ The names of tables and table columns must start with a letter, and no arithmetic operators are permitted ■ Reading and writing with SQL functions	■ Freely definable tables (.TAB files) ■ Reading and writing with FN functions
Traverse in tool-axis direction		
■ Manual operation (3-D ROT menu)	■ X	■ X, FCL2 function
■ With handwheel superimpositioning	■ X	■ X, option 44
Entry of feed rates:		
■ **FT** (time in seconds for path)	■ –	■ X
■ **FMAXT** (only for active rapid traverse potentiometer: time in seconds for path)	■ –	■ X
FK free contour programming		
■ Conversion of FK program to Klartext conversational language	■ –	■ X
■ FK blocks in combination with **M89**	■ –	■ X
Program jumps:		
■ Maximum number of labels	■ 65535	■ 1000
■ Subprograms	■ X	■ X
■ Nesting depth for subprograms	■ 20	■ 6
Q parameter programming:		
■ FN 15: PRINT	■ –	■ X
■ FN 25: PRESET	■ –	■ X
■ FN 29: PLC LIST	■ X	■ –
■ FN31: RANGE SELECT	■ –	■ X
■ FN32: PLC PRESET	■ –	■ X
■ FN37: EXPORT	■ X	■ –
■ Write to LOG file with **FN16**	■ X	■ –
■ Displaying parameter contents in the additional status display	■ X	■ –
■ **SQL** functions for writing and reading tables	■ X	■ –

Function	TNC 640	iTNC 530
Graphic support		
▪ 2-D programming graphics	▪ X	▪ X
▪ REDRAW function (**REDRAW**)	▪ –	▪ X
▪ Show grid lines as the background	▪ X	▪ –
▪ Test graphics (plan view, projection on 3 planes, 3-D view)	▪ X	▪ X
▪ Coordinates of line intersection for projection in 3 planes	▪ –	▪ X
▪ Factor in tool change macro	▪ X (differing to actual execution)	▪ X
Preset table		
▪ Line 0 of the preset table can be edited manually	▪ X	▪ –
Programming aids:		
▪ Color highlighting of syntax elements	▪ X	▪ –
▪ Calculator	▪ X (scientific)	▪ X (standard)
▪ Convert NC blocks to comments	▪ X	▪ –
▪ Structure blocks in NC program	▪ X	▪ X
▪ Structure view in test run	▪ –	▪ X
Dynamic Collision Monitoring (DCM):		
▪ Fixture monitoring	▪ –	▪ X, option 40
▪ Tool carrier management	▪ X	▪ X, option 40

Function	TNC 640	iTNC 530
CAM support:		
■ Load contours from Step data and Iges data	■ X, option 42	■ –
■ Load machining positions from Step data and Iges data	■ X, option 42	■ –
■ Offline filter for CAM files	■ –	■ X
■ Stretch filter	■ X	■ –
MOD functions:		
■ User parameters	■ Config data	■ Numerical structure
■ OEM help files with service functions	■ –	■ X
■ Data medium inspection	■ –	■ X
■ Load service packs	■ –	■ X
■ Specify the axes for actual position capture	■ –	■ X
■ Configure counter	■ X	■ –
Special functions:		
■ Create reverse program	■ –	■ X
■ Define the counter with **FUNCTION COUNT**	■ X	■ –
■ Define the dwell time with **FUNCTION FEED**	■ X	■ –
■ Define the dwell time with **FUNCTION DWELL**	■ X	■ –
■ Determine the integration of the programmed coordinates with **FUNCTION PROG PATH**	■ X	■ –
Status displays:		
■ Dynamic display of Q-parameter contents, definable number ranges	■ X	■ –
■ Graphic display of residual run time	■ –	■ X
Individual color settings of user interface	–	X

Comparison: Touch probe cycles in the Manual operation and Electronic handwheel operating modes

Cycle	TNC 640	iTNC 530
Touch-probe table for managing 3-D touch probes	X	–
Calibrating the effective length	X	X
Calibrating the effective radius	X	X
Measuring a basic rotation using a line	X	X
Setting the preset on any axis	X	X
Setting a corner as preset	X	X
Setting a circle center as preset	X	X
Setting a center line as preset	X	X
Measuring a basic rotation using two holes/cylindrical studs	X	X
Setting the preset using four holes/cylindrical studs	X	X
Setting the circle center using three holes/cylindrical studs	X	X
Determine and offset misalignment of a plane	X	–
Support of mechanical touch probes by manually capturing the current position	By soft key or hard key	By hard key
Write measurement values to the preset table	X	X
Write measurement values to the datum table	X	X

Comparison: Differences in programming

Function	TNC 640	iTNC 530
File management:		
■ Entry of name	■ Opens **Select file** pop-up window**Select file**	■ Synchronizes the cursor
■ Support of key combinations	■ Not available	■ Available
■ Favorites Management	■ Not available	■ Available
■ Configuration of column structure	■ Not available	■ Available
Selecting a tool from the table	Selection via split-screen menu	Selection in a pop-up window
Programming special functions with the **SPEC FCT** key	Pressing the key opens a soft-key row as a submenu. To exit the submenu, press the **SPEC FCT** key again; then the control shows the last active soft-key row	Pressing the key adds the soft-key row as the last row. To exit the menu, press the **SPEC FCT** key again; then the control shows the last active soft-key row
Programming approach and departure motions with the **APPR DEP** key	Pressing the key opens a soft-key row as a submenu. To exit the submenu, press the **APPR DEP** key again; then the control shows the last active soft-key row	Pressing the key adds the soft-key row as the last row. To exit the menu, press the **APPR DEP** key again; then the control shows the last active soft-key row
Pressing the hard key **END** with active **CYCLE DEF** and **TOUCH PROBE** menus	Terminates the editing process and calls the file manager	Exits the respective menu
Calling the file manager if **CYCLE DEF** and **TOUCH PROBE** menus are active	Terminates the editing process and calls the file manager. The respective soft-key row remains selected when the file manager is exited	**Key non-functional** error message
Calling the file manager if **CYCL CALL**, **SPEC FCT**, **PGM CALL** and **APPR DEP** menus are open	Terminates the editing process and calls the file manager. The respective soft-key row remains selected when the file manager is exited	Terminates the editing process and calls the file manager. The basic soft-key row is selected when the file manager is exited

Function	TNC 640	iTNC 530
Datum table:		
■ Sorting function by values within an axis	■ Available	■ Not available
■ Resetting the table	■ Available	■ Not available
■ Switching the list/form view	■ Switch via the screen layout key	■ Switchover by toggle soft key
■ Inserting individual line	■ Allowed everywhere, renumbering possible after request. Empty line is inserted, must be filled with zeros manually	■ Only allowed at the end of the table. Line with value 0 in all columns is inserted
■ Transfer of actual position values on individual axis to the datum table using the keys	■ Available in the **Program Run Single Block** and **Program Run, Full Sequence** operating modes	■ Available
■ Transfer of actual position values on all active axes to the datum table using the keys	■ Not available	■ Available
■ Capturing the last positions measured by TS using the keys	■ Not available	■ Available
FK free contour programming:		
■ Programming of parallel axes	■ With X/Y coordinates, independent of machine type; switchover with **FUNCTION PARAXMODE**	■ Machine-dependent with the existing parallel axes
■ Automatic correction of relative references	■ Relative references in contour subprograms are not corrected automatically	■ All relative references are corrected automatically
■ Specify the working plane during programming	■ BLK form ■ **Plane XY ZX YZ** soft key if the working plane differs	■ BLK form
Q-parameter programming:		
■ Q-parameter formula with SGN	Q12 = SGN Q50 ■ if Q 50 = 0 then Q12 = 0 ■ if Q50 > 0 then Q12 = 1 ■ if Q50 < 0 then Q12 = -1	Q12 = SGN Q50 ■ if Q50 >= 0 then Q12 = 1 ■ if Q50 < 0 then Q12 = -1

Function	TNC 640	iTNC 530
Handling of error messages:		
■ Help with error messages	■ Call via **ERR** key	■ Call via **HELP** key
■ Switching the operating mode while help menu is active	■ Help menu is closed when the operating mode is switched	■ Operating mode switchover is not allowed (key is non-functional)
■ Selecting the background operating mode while help menu is active	■ Help menu is closed when F12 is used for switching	■ Help menu remains open when F12 is used for switching
■ Identical error messages	■ Are collected in a list	■ Are displayed only once
■ Acknowledgment of error messages	■ Every error message (even if it is displayed more than once) must be acknowledged, the **DELETE ALL** function is available	■ Error message to be acknowledged only once
■ Access to protocol functions	■ Log and powerful filter functions (errors, keystrokes) are available	■ Complete log without filter functions available
■ Saving service files	■ Available. No service file is generated upon a system crash ■ Error number selectable for which a service file will be generated automatically	■ Available. A service file is generated automatically upon a system crash
Find function:		
■ List of words recently searched for	■ Not available	■ Available
■ Show elements of active block	■ Not available	■ Available
■ Show list of all available NC blocks	■ Not available	■ Available
Starting the search function with the up/down arrow keys when highlighted	Works up to max. 100000 NC blocks, can be set via configuration datum	No limitation regarding program length
Programming graphics:		
■ True-to-scale display of grid	■ Available	■ Not available
■ Editing contour subprograms in SLII cycles with **AUTO DRAW ON**	■ With error messages, in the main program the cursor is positioned on the **CYCL CALL** NC block	■ With error messages, the cursor is positioned on the NC block in the contour subprogram that caused the error
■ Moving the zoom window	■ Repeat function not available	■ Repeat function available

Function	TNC 640	iTNC 530
Programming minor axes:		
▪ Syntax **FUNCTION PARAXCOMP**: Define the behavior of the display and the paths of traverse	▪ Available	▪ Not available
▪ Syntax **FUNCTION PARAXMODE**: Define the assignment of the parallel axes to be traversed	▪ Available	▪ Not available
Programming OEM cycles		
▪ Access to table data	▪ Via **SQL** commands and via **FN 17** and **FN 18** or **TABREAD** / **TABWRITE** functions ▪	▪ Via **FN 17** and **FN 18** or **TABREAD** / **TABWRITE** functions
▪ Access to machine parameters	▪ With the **CFGREAD** function	▪ Via **FN 18** functions
▪ Creating interactive cycles with **CYCLE QUERY**, e.g. touch probe cycles in Manual Operation	▪ Available	▪ Not available

Comparison: Differences in Test Run, functionality

Function	TNC 640	iTNC 530
Entering a program with the **GOTO** key	Function only possible if the **START SINGLE** soft key was not pressed	Function also possible after **START SINGLE**
Calculation of machining time	Each time the simulation is repeated by pressing the START soft key, the machining time is totaled	Each time the simulation is repeated by pressing the START soft key, time calculation starts at 0
Single block	With point pattern cycles and **CYCL CALL PAT**, the control stops after each point	Point pattern cycles and **CYCL CALL PAT** are handled by the control as a single NC block

Comparison: Differences in Test Run, operation

Function	TNC 640	iTNC 530
Zoom function	Each sectional plane can be selected by individual soft keys	Sectional plane can be selected via three toggle soft keys
Machine-specific miscellaneous functions M	Lead to error messages if they are not integrated in the PLC	Are ignored during Test Run
Displaying/editing the tool table	Function available via soft key	Function not available
Tool depiction	■ Turquoise: Tool length ■ Red: Length of cutting edge and tool is engaged ■ Blue: Length of cutting edge and tool is not engaged	■ - ■ Red: Tool is engaged ■ Green: Tool is not engaged
View options of 3-D view	Available	Function not available
Adjustable model quality	Available	Function not available

Comparison: Differences in Manual Operation, functionality

Function	TNC 640	iTNC 530
Jog increment function	The jog increment can be defined separately for linear and rotary axes	The jog increment applies for both linear and rotary axes
Preset table	Basic transformation (translation and rotation) from machine table system to workpiece system via the **X**, **Y**, and **Z** columns, as well as spatial angles **SPA**, **SPB** and **SPC**. In addition, the columns **X_OFFS** to **W_OFFS** can be used to define the axis offset of each individual axis. The function of the axis offsets can be configured. Line 0 can also be edited manually.	Basic transformation (translation) from machine table system to workpiece system via the **X**, **Y** and **Z** columns, as well as a basic rotation **ROT** in the working plane (rotation). In addition, the columns **A** to **W** can be used to define datums in the rotary and parallel axes. Line 0 can only be edited by manual probing cycles.
Behavior when presetting	Presetting in a rotary axis has the same effect as an axis offset. This offset is also effective for kinematics calculations and during tilting of the working plane. In machine parameter **presetToAlignAxis** (no. 300203) your machine tool builder specifies for each axis what effect an offset of a rotational axis has on the preset. ■ **True** (Default): Use offset for aligning the workpiece ■ **False**: Use offset for inclined-tool machining	Rotary axis offsets defined by machine parameters do not influence the axis positions that were defined in the Tilt working plane function. MP7500, bit 3 permits you to define whether the current rotary axis position relative to the machine datum is taken into account or whether the first rotary axis (usually the C axis) is assumed to be in 0° position.
Presetting	Only after a reference run is it possible to set a preset or to modify a preset via the preset table.	A preset can be set or modified via the preset table before a reference run.
Handling of the preset table:		
Defining the feed rates	Feed rates can be defined separately for linear and rotary axes By pressing the **F** soft key in the **Manual operation** mode, you can define separate feed rates for linear and rotary axes. These feed rates apply only in the **Manual operation** mode.	Only one feed-rate can be defined for linear and rotary axes

Comparison: Differences in Manual Operation, operation

Function	TNC 640	iTNC 530
Capturing the position values from mechanical probes	Confirm actual position with a soft key or hard key	Actual-position capture by hard key

Comparison: Differences in Program Run, operation

Function	TNC 640	iTNC 530
Operating mode switchover after program run has been suspended by switching to the **Program run, single block** operating mode and canceled with **INTERNAL STOP**	When you return to the **Program run, full sequence** mode: error message **Current block not selected**. Use block scan to select the point of interruption	Switching the operating mode is allowed, modal information is saved, program run can be continued by pressing NC start
GOTO is used to go to FK sequences after program run was interrupted there before switching the operating mode	Error message **FK programming: Undefined starting position** Entering with mid-program startup is permitted	GOTO allowed
Mid-program startup:		
Switching the screen layout for mid-program startup	Only possible, if startup position has already been approached	Possible in all operating states
Error messages	Error messages are still active after the error has been corrected and must be acknowledged separately	Error messages are sometimes acknowledged automatically after the error has been corrected
Point patterns in single block	With point pattern cycles and **CYCL CALL PAT**, the control stops after each point.	Point pattern cycles and **CYCL CALL PAT** are handled by the control as a single NC block

Comparison: Differences in Program Run, traverse movements

NOTICE
Danger of collision!
NC programs that were created older controls can lead to unexpected axis movements or error messages on current control models. Danger of collision during machining!
▶ Check the NC program or program section using the graphic simulation
▶ Carefully test the NC program or program section in the **Program run, single block** operating mode
▶ Pay attention to the following known differences (the list below might not be complete!)

Function	TNC 640	iTNC 530
Handwheel-superimposed traversing with **M118**	Effective in the machine coordinate system If the Global Program Settings option is active, M118 is in effect in the coordinate system selected most recently for handwheel superimpositioning.	Effective in the machine coordinate system
Deleting basic rotation with **M143**	**M143** clears the entries from the **SPA**, **SPB** and **SPC** columns in the preset table	**M143** does **not** delete the entry in the **ROT** column in the preset table; but only in the NC program; reactivating the corresponding preset table line activates the deleted basic rotation
Scaling approach/departure movements (**APPR/DEP/RND**)	Axis-specific scaling factor is allowed, radius is not scaled	Error message
Approach/departure with **APPR/DEP**	Error message if **R0** is programmed for **APPR/DEP LN** or **APPR/DEP CT**	Tool radius 0 and compensation direction **RR** are assumed
Approach/departure with **APPR/DEP** if contour elements with length 0 are defined	Contour elements with length 0 are ignored. The approach/departure movements are calculated for the first and last valid contour element	An error message is issued if a contour element with length 0 is programmed after the **APPR** block (relative to the first contour point programmed in the APPR block) For a contour element with length 0 before a **DEP** block, the iTNC 530 does not issue an error message, but uses the last valid contour element to calculate the departure movement

Function	TNC 640	iTNC 530
Effect of Q parameters	**Q60** to **Q99** (**QS60** to **QS99**) are always local.	**Q60** to **Q99** (**QS60** to **QS99**) are local or global, depending on MP7251 in converted cycle programs (.cyc). Nested calls may cause problems
Automatic cancelation of tool radius compensation	◾ NC block with **R0** ◾ **DEP** block ◾ Program selection ◾ **END PGM**	◾ NC block with **R0** ◾ **DEP** block ◾ Program selection ◾ Programming of Cycle 10 **ROTATION** ◾ **PGM CALL**
NC blocks with **M91**	No consideration of tool radius compensation	Consideration of tool radius compensation
Behavior with **M120 LA1**	No effect on processing, as the control interprets the input internally as an **LA0**	Possible undesired effect on processing, as the control interprets the entry internally as an **LA2**
Block scan in a point table	The tool is positioned above the next position to be machined	The tool is positioned above the last position that has been completely machined
Emtpy **CC** block (pole adoption from last tool position) in the NC program	Last positioning block in the working plane must contain both coordinates of the working plane	Last positioning block in the working plane does not necessarily need to contain both coordinates of the working plane. Can cause problems with **RND** or **CHF** blocks
Axis-specific scaling of **RND** block	**RND** block is scaled, the result is an ellipse	Error message is issued
Reaction if a contour element with length 0 is defined before or after a **RND** or **CHF** block	Error message is issued	Error message is issued if a contour element with length 0 is located before the **RND** or **CHF** block Contour element with length 0 is ignored if the contour element with length 0 is located after the **RND** or **CHF** block

Function	TNC 640	iTNC 530
Circle programming with polar coordinates	The incremental rotation angle **IPA** and the direction of rotation **DR** must have the same sign. Otherwise, an error message will be issued	The algebraic sign of the direction of rotation is used if the sign defined for **DR** differs from the one defined for **IPA**
Tool radius compensation on circular arc or helix with angular length = 0	The transition between the adjacent elements of the arc/helix is generated. Also, the tool axis motion is executed right before this transition. If the element is the first or last element to be corrected, the next or previous element is dealt with in the same way as the first or last element to be corrected	The equidistant line of the arc/helix is used for generating the tool path

SLII Cycles 20 to 24:

	TNC 640	iTNC 530
■ Number of definable contour elements	■ Max. 16384 blocks in up to 12 subcontours	■ Max. 8192 contour elements in up to 12 subcontours, no restrictions for subcontour
■ Define the working plane	■ Tool axis in **TOOL CALL** block defines the working plane	■ The axes of the first positioning block in the first subcontour define the working plane
■ Position at end of SL cycle	■ With the **posAfterContPocket** (no. 201007) parameter, you can define whether the end position is above the last programmed position, or whether the tool moves to clearance height in the tool axis ■ If the tool moves to clearance height in the tool axis, both coordinates must be programmed with the first traverse movement	■ With MP7420, you can define whether the end position is above the last programmed position, or whether the tool moves only to clearance height in the tool axis ■ If the tool moves to clearance height in the tool axis, one coordinate must be programmed with the first traverse movement

Function	TNC 640	iTNC 530
SLII Cycles 20 to 24:		
▪ Behavior with islands not contained in pockets	▪ Cannot be defined with complex contour formula	▪ Restricted definition in complex contour formula is possible
▪ Set operations for SL cycles with complex contour formulas	▪ Real set operation possible	▪ Only restricted performance of real set operation possible
▪ Radius compensation is active during **CYCL CALL**	▪ Error message is issued	▪ Radius compensation is canceled, NC program is being run
▪ Paraxial positioning blocks in contour subprogram	▪ Error message is issued	▪ NC program is being run
▪ Miscellaneous functions **M** in contour subprogram	▪ Error message is issued	▪ M functions are ignored
Cylinder surface machining in general:		
▪ Contour definition	▪ With X/Y coordinates, independent of machine type	▪ Machine-dependent, with existing rotary axes
▪ Offset definition on cylinder surface	▪ With datum shift in X/Y, regardless of machine type	▪ Machine-specific datum shift in rotary axes
▪ Offset definition for basic rotation	▪ Function available	▪ Function not available
▪ Circle programming with C/CC	▪ Function available	▪ Function not available
▪ **APPR/DEP** blocks in contour definition	▪ Function not available	▪ Function available
Cylinder surface machining with Cycle 28:		
Complete roughing-out of slot	Function available	Function not available
Cylinder surface machining with Cycle 29	Direct plunging to contour of ridge	Circular approach to contour of ridge
Cycles 25x for pockets, studs and slots:		
▪ Plunging movements	In limit ranges (geometrical conditions of tool/contour) error messages are triggered if plunging movements lead to unreasonable/critical behavior	In limit ranges (geometrical conditions of tool/contour), vertical plunging is used if required

Function	TNC 640	iTNC 530
PLANE function:		
■ **TABLE ROT/COORD ROT**	Effect: ■ The transformation types are effective on all free rotary axes ■ The control does not always position the free rotary axis with **TABLE ROT**, but depending on the current position, the programmed spatial angle and the machine kinematics Default with missing selection: ■ **COORD ROT** is used	Effect ■ The transformation types are only effective with a C rotary axis ■ With **TABLE ROT** the control always positions the rotary axis Default with missing selection: ■ **COORD ROT** is used
■ Positioning behavior	■ **SYM** ■ **SEQ**	■ **SEQ**
■ Machine is configured for axis angle	■ All **PLANE** functions can be used	■ Only **PLANE AXIAL** is executed
■ Programming an incremental spatial angle according to **PLANE AXIAL**	■ Error message is issued	■ Incremental spatial angle is interpreted as an absolute value
■ Programming an incremental axis angle according to **PLANE SPATIAL** if the machine is configured for spatial angles	■ Error message is issued	■ Incremental axis angle is interpreted as an absolute value
■ Programming of **PLANE** functions with active Cycle 8 **MIRROR IMAGE**	■ Mirroring has no influence on tilting using **AXIAL PLANE** and Cycle **19**	■ Function is available with all **PLANE** functions
■ Axis positioning on machines with two rotary axes e.g. **L A+0 B+0 C+0** or **L A+Q120 B+Q121 C+Q122**	■ Only possible after a tilting function (error message if without a tilting function) ■ Parameters that are not defined are given the status **UNDEFINED**; they are not given the value 0	■ Possible at any time if spatial angles are used (machine parameter setting) ■ The control assigns the value 0 to parameters that are not defined
Special functions for cycle programming:		
■ FN 17	■ Values are always output in metric form	■ Values are output in the units of the active NC program
■ FN 18	■ Values are always output in metric form	■ Values are output in the unit of the active NC program
Compensation of tool length in the position display	The tool length entries **L** and **DL** from the tool table are taken into account in the position display, from **TOOL CALL** block depending on the machine parameter **progToolCallDL**(no. 124501)	The tool length entries **L** and **DL** from the tool table are taken into account in the position display

Comparison: Differences in MDI operation

Function	TNC 640	iTNC 530
Miscellaneous functions	■ Status display for Q parameters ■ Block functions, e.g. **COPY BLOCK** ■ ACC setting ■ Program functions for turning ■ Miscellaneous program functions, e.g. **FUNCTION DWELL**	
Skip NC blocks	Separate soft key for MDI mode	The soft key from the **Program Run Full Sequence** operating mode is effective

Comparison: Differences in programming station

Function	TNC 640	iTNC 530
Demo version	NC programs with more than 100 NC blocks cannot be selected; an error message is issued	NC programs can be selected, max. 100 NC blocks are displayed, further NC blocks are truncated in the display
Demo version	If nesting with **PGM CALL** results in more than 100 NC blocks, there is no test graphic display; an error message is not issued	Nested NC programs can be simulated
Demo version	You can transfer up to 10 elements from the CAD viewer to an NC program.	You can transfer up to 31 lines from the DXF converter to an NC program.
Copying NC programs	Copying to and from the directory **TNC:** is possible with Windows Explorer	**TNCremo** or file manager of programming station must be used for copying
Shifting the horizontal soft-key row	Clicking on the soft-key bar shifts one soft-key row to the right or left	Clicking any soft-key bar activates the respective soft-key row

Index

HEIDENHAIN

www.klartext-portal.com
The Information Site for
HEIDENHAIN Controls

Klartext App
The Klartext on Your
Mobile Device

Google   Apple
Play Store   App Store

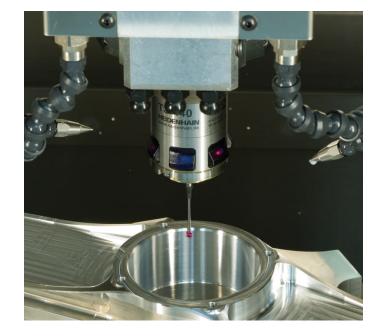# Touch probes from HEIDENHAIN

help you reduce non-productive time and improve the
dimensional accuracy of the finished workpieces.

Workpiece touch probes

TS 220   Signal transmission by cable
TS 440   Infrared transmission
TS 642, TS 740   Infrared transmission

- Workpiece alignment
- Preset setting
- Workpiece measurement

Tool touch probes

TT 160   Signal transmission by cable
TT 460   Infrared transmission

- Tool measurement
- wear monitoring
- tool breakage detection