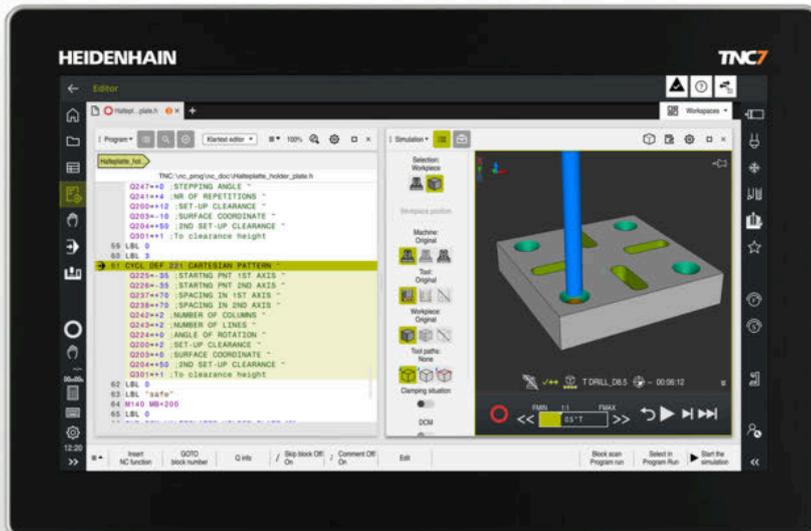# HEIDENHAIN

TNC7 go

User's Manual
Setup and Program Run

NC Software
81762x-20

Table of contents

1

**About the
User's Manual**

1.1 Target group: Users

A user is anyone who uses the control to perform at least one of the following tasks:
- Operating the machine
  - Setting up tools
  - Setting up workpieces
  - Machining workpieces
  - Eliminating possible errors during program run
- Creating and testing NC programs
  - Creating NC programs at the control
  - Using the Simulation mode to test the NC programs
  - Eliminating possible errors during program test

The depth of information in the User's Manual results in the following qualification requirements on the user:
- Basic technical understanding (e.g., spatial imagination and the ability to read technical drawings)
- Basic knowledge in the field of metal cutting (e.g., understanding the meaning of material-specific parameters)
- Safety instructions (e.g., understanding possible dangers and how to avoid them)
- Training on the machine (e.g., comprehending axis directions and the machine configuration)

> **i** HEIDENHAIN offers separate information products for other target groups:
> - Leaflets and overview of the product portfolio for potential buyers
> - Service Manual for service technicians
> - Technical Manual for machine manufacturers
>
> Additionally, HEIDENHAIN provides users and lateral entrants with a wide range of training opportunities in the field of NC programming.
> **HEIDENHAIN training portal**

In line with the target group, this User's Manual only contains information on the operation and use of the control. The information products for other target groups contain information on further product life phases.

1.2 Available user documentation

User's Manual

HEIDENHAIN refers to this information product as a User's Manual, regardless of the output or transport medium. Well-known designations with the same meaning include operator's manual and operating instructions.

The User's Manual for the control is available in the variants below:

- As a PDF file containing a **complete edition** of the User's Manual with all contents
  ID: 1441440-xx

  https://content.heidenhain.de/doku/tnc_guide/html/en/

- As an HTML file with all contents for use as the integrated product aid **TNCguide**: directly on the control

  https://content.heidenhain.de/doku/tnc_guide/html/en/

- As a printed version, sub-divided into the modules below:
  - The **Setup and Program Run** User's Manual contains all information needed for setting up the machine and for executing NC programs.
    ID: 1441439-xx
  - The **Programming and Testing** User's Manual contains all information needed for creating and testing NC programs.
    ID: 1441438-xx

The User's Manual supports you in the safe handling of the control according to its intended use.

Further information: "Proper and intended use", Page 42

📖 **Complete edition and TNCguide for the TNC7 go**

Binding of the printed version of the User's Manual is only possible if a certain number of pages is not exceeded.

Die **complete edition** and **TNCguide** may include additional sample programs for NC functions that are not contained in the printed version.

You can download this documentation free of charge from the HEIDENHAIN website.

Further information products for users

The following information products are available:

- The **Overview of New and Modified Software Functions** informs you about the innovations of specific software versions.

  https://content.heidenhain.de/doku/tnc_guide/html/en/

- **Overview of the machine parameters, error numbers and system data**, providing the following functions:
  - Machine parameters of the **MPs for setters** application
  - Preassigned error numbers of the **FN 14: ERROR** NC function (ISO: **D14**)
  - System data readable with the **FN 18: SYSREAD** (ISO: **D18**) and **SYSSTR** NC functions

  https://content.heidenhain.de/doku/tnc_guide/html/en/

- **HEIDENHAIN brochures** inform you about products and services from HEIDENHAIN (e.g., software options of the control).

  HEIDENHAIN brochures

- The **NC Solutions** database offers solutions for frequently occurring tasks.

  HEIDENHAIN NC solutions

1.3 Types of notes used

Safety precautions

Comply with all safety precautions indicated in this document and in your machine manufacturer'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*
>
> **Caution** 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 important additional or supplementary information.

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

The book symbol indicates a **cross reference**.
A cross reference leads to external documentation, for example the documentation of your machine manufacturer or other supplier.

1.4 Notes on using NC programs

NC programs contained in this User's Manual are suggestions for solutions. The NC programs or individual NC blocks must be adapted before being used on a machine.

Change the following contents as needed:

- Tools
- Cutting parameters
- Feed rates
- Clearance height or safe position
- Machine-specific positions (e.g., with **M91**)
- Paths of program calls

Some NC programs depend on the machine kinematics. Adapt these NC programs to your machine kinematics before the first test run.

In addition, test the NC programs using the simulation before the actual program run.

> **i** With a program test you determine whether the NC program can be used with the available software options, the active machine kinematics and the current machine configuration.

1.5 User's Manual as integrated product aid: TNCguide

Application

The integrated product aid **TNCguide** offers the full content of all User's Manuals.

Further information: "Available user documentation", Page 31

The User's Manual supports you in the safe handling of the control according to its intended use.

Further information: "Proper and intended use", Page 42

Related topics

- The **Help** workspace

  Further information: Programming and Testing User's Manual

Requirement

In the factory default setting, the control offers the integrated product aid **TNCguide** in German and English language versions.

If the control cannot find a **TNCguide** language version matching the selected dialog language, it opens **TNCguide** in English.

If the control cannot find a **TNCguide** language version, it opens an information page with instructions. With the link available there and the steps provided, you can supplement the files missing in the control.

> **i** You can also open the information page manually by selecting the **index.html** file (for example, at **TNC:\tncguide\en\readme**). The path depends on the desired language version (e.g., **en** for English).
>
> With the steps provided you can also update the **TNCguide** version. Updating may be required (e.g., after a software update).

Description of function

The integrated product aid **TNCguide** can be selected within the **Help** application or in the **Help** workspace.

Further information: "The Help application", Page 36

Further information: Programming and Testing User's Manual

Operation of **TNCguide** is identical in both cases.

Further information: "Icons", Page 37

The Help application
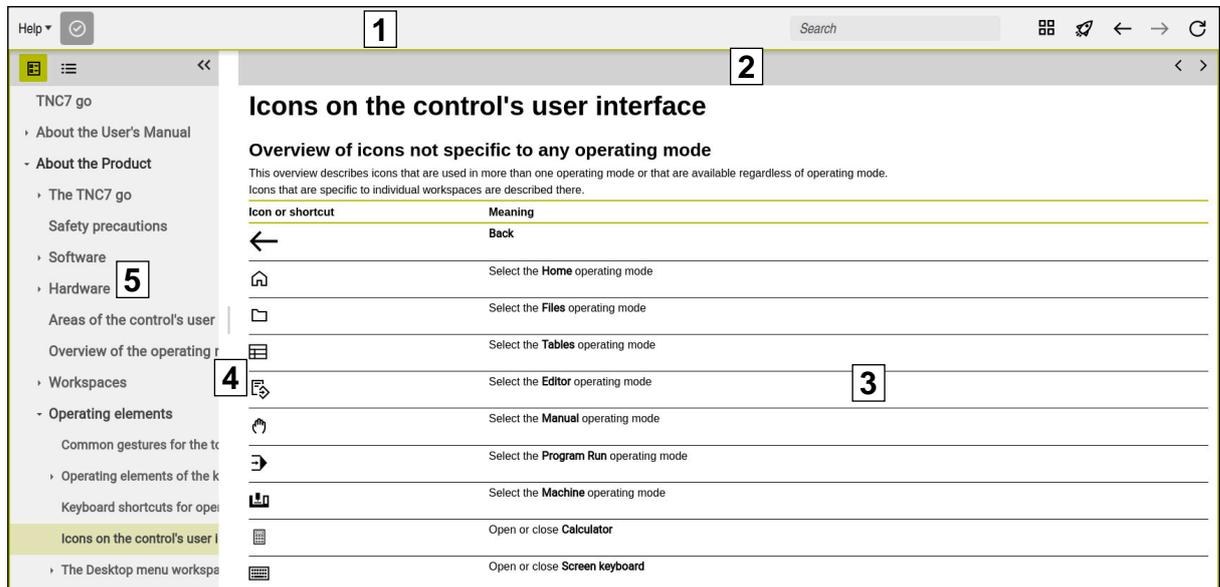
Open **TNCguide** in the **Help** workspace

TNCguide includes the following areas:

1   Title bar of the **Help** workspace
    Further information: "The Help workspace", Page 37
2   Title bar of the integrated product aid **TNCguide**
    Further information: "TNCguide ", Page 37
3   Content column of the **TNCguide**
4   Separator between the columns of the **TNCguide**
    Adjust the column width by means of the separator.
5   Navigation column of the **TNCguide**

Icons

The Help workspace

The **Help** workspace within the **Help** application includes the following icons:

Icon	Meaning
⊘	Open or close the **Search results** column
	Further information: "Searching in TNCguide", Page 38
⊞	**Open Home page**
	The start page displays all available documentation. Select the desired documentation using navigation tiles (e.g., **TNCguide**).
	If only one piece of documentation is available, the control opens the content directly.
	When a documentation is open, you can use the search function.
⟋	**Open Tutorials**
← →	**Navigate**
	Navigate between the contents opened recently
↻	**Refresh**

TNCguide

The integrated **TNCguide** product aid includes the following icons:

Icon	Meaning
⊟	**Open Structure**
	The structure consists of the content headings.
	The structure serves for main navigation within the documentation.
☰	**Open Index**
	The index consists of important keywords.
	The index serves as an alternative navigation within the documentation.
‹ ›	**Navigate**
	Display previous or next page within the documentation
« »	**Open or close**
	Display or hide the navigation
⧉	**Copy**
	Copy NC examples to the clipboard
	Further information: "Copying NC examples to clipboard", Page 39

Context-sensitive help

You can open **TNCguide** for the current context. Context-sensitive help means that the relevant information is displayed directly (e.g., for the selected item or the current NC function).

To call context-sensitive help, the following elements are available:

Icon or key	Meaning
⑦	**Help** icon If you select the icon and then one of the items in the user interface, the control will open the associated information in **TNCguide**.
HELP	**HELP** key If you press the **HELP** key while editing an NC block, the control will display the associated information in **TNCguide**.

If you call TNCguide in a certain context, the control opens the contents in a pop-up window. If you select the **Show more** button, the control will open **TNCguide** in the **Help** application.

Further information: "The Help application", Page 36

If the **Help** workspace is already open, the control displays **TNCguide** there and will not open a pop-up window.

Further information: Programming and Testing User's Manual

1.5.1 Searching in TNCguide

Using the search function, you can search for the entered search terms within the open documentation.

Use the search function as follows:

▶ Enter a string in **Search**

> 🛈 The search starts automatically after you enter a character.
> If you wish to delete the entry, use the X symbol within the entry field.

> The control opens the column containing the search results.
> The control marks references also within open content pages.
▶ Select the reference
> The control opens the selected content.
> The control continues displaying the results of the last search.
▶ Select an alternative reference if necessary
▶ Enter a new character string if required

1.5.2 Copying NC examples to clipboard

Use the copy function to copy NC examples from the documentation to the NC editor.

To use the copy function:

▶ Navigate to the desired NC example

▶ Expand **Notes on using NC programs**

▶ Read and follow **Notes on using NC programs**

   Further information: "Notes on using NC programs", Page 34

▶ Copy NC example to clipboard

> The button switches colors while copying.

> The clipboard contains the entire content of the copied NC example.

▶ Insert the NC example into the NC program

▶ Adapt the inserted content according to the **Notes on using NC programs**

▶ Use the Simulation mode to test the NC program

   Further information: Programming and Testing User's Manual

1.6 Contacting the editorial staff

Would you like any changes, or have you found any errors?

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.com

2

About the product

2.1 The TNC7 go

Every HEIDENHAIN control supports you with dialog-guided programming and finely detailed simulation. The TNC7 go additionally offers you form-based programming to reach the desired result quickly and easily.

Software options and optional hardware extensions can be used for flexibly increasing the range of functions and ease of use.

Operation is made easier, for example, by using touch probes, handwheels or a 3D mouse.

Further information: "Hardware enhancements", Page 56

Definitions

Abbreviation	Definition
TNC	**TNC** is derived from the acronym **CNC** (computerized numerical control). The **T** (tip or touch) stands for the capability of entering NC programs directly at the control or to program them graphically using gestures.
7	The product number indicates the control generation. The range of functions depends on the enabled software options.
go	The addition go shows that the control is a workshop-oriented contouring control that enables you to program common milling and drilling operations right at the machine.

2.1.1 Proper and intended use

The information about proper and intended use supports you in safely handling a product such as a machine tool.

The control is a machine component but not a complete machine. This User's Manual describes the use of the control. Before using the machine including the control, take the OEM documentation to inform yourself about the safety-related aspects, the necessary safety equipment as well as the requirements on the qualified personnel.

> **i** HEIDENHAIN sells controls designed for milling and turning machines as well as for machining centers with up to 24 axes. If you as a user face a different constellation, then contact the owner immediately.

HEIDENHAIN contributes additionally to enhancing your safety and that of your products, notably by taking customer feedback into consideration. This results, for example, in function adaptations of the controls and safety precautions in the information products.

> **i** Contribute actively to increasing the operational safety by reporting any missing or misleading information.
> **Further information:** "Contacting the editorial staff", Page 40

2.1.2 Data Architecture and Access Options

Directories

While in use, the control will store various types of data on its internal data carrier.

Partition or directory	Data
root	Operating system
/mnt/tnc/	Process, application, and user data
	Examples: NC programs, tables or 3D models
/mnt/plc/	Machine manufacturer data
	Examples: PLC program, machine configuration or additional software
/mnt/sys/	NC software

The type, format, and amount of data stored depend on various factors. For example, the software options that are available and actually used as well as the functions used, including the current settings, have a great influence here.

Data acquisition

While in use, the control will generate different pieces of data. Depending on the functions in use, for example, data acquisition is performed continuously and in real time.

Data acquisition examples:

- Backup

  You can manually create a backup copy of the TNC partition in order to avoid potential loss of data.

- Service file

  In the event of a fault, you can manually create a service file that supports troubleshooting by a service technician.

- User administration

  If user administration is active, the control assigns defined roles and permissions to the logged-on users. When a user logs on, the control will acquire personal or anonymous data in connection with its use.

- Tool-usage file

  Using tool-usage files, the control can, for example, check whether the tool's remaining life is sufficient for the current NC program.

- Log functions or log files (e.g., from touch-probe cycles)

  Using various log files, you can, for example, check and compare the results of touch-probe cycles.

  The control needs other function-dependent log files to, for example, detect process variations.

> **i** For more details on data acquisition, please refer to the sections describing the individual functions and the **Settings** application.

Data transfer and data access

Machine manufacturers, machine operators, or users can transfer data to external data carriers or drives, for example, and set up and configure a data transfer functionality. The user has control over and responsibility for the stored data (e.g., with respect to safety and the duration of storage).

Examples of data transfer:

- **HEIDENHAIN DNC** (#18 / #3-03-1)

  This software option enables external Windows applications to access data of the control via the TCP/IP protocol.

- **OPC UA NC Server** (#56-61 / #3-02-1*)

  These software options include OPC UA, a standardized interface that allows external access to the data and functions of your control.

- USB devices

  USB devices can be used to transfer data or make external backups.

> **ⓘ** For more details on data transfer, please refer to the sections describing the individual functions and the **Settings** application.

Data access via file management:

- Drives, folders, and files

  You can access external data or make external backups by using connected drives or USB devices. For example, you can manage, create, edit, or delete folders and files.

- Functions and settings

  In the file manager you can search for certain contents or display hidden or dependent files.

> **ⓘ** For further details on file access, please refer to the section on the **Files** operating mode.
>
> Please observe the safety instructions and information given by the machine manufacturer in order to avoid loss of data or jeopardize the functionality of your system.

2.1.3 Intended place of operation

In accordance with the DIN EN 50370-1 standard for electromagnetic compatibility (EMC), the control is approved for use in industrial environments.

Definitions

Guideline	Definition
DIN EN 50370-1:2006-02	This standard deals, among other things, with interference emissions and immunity to interference of machine tools.

2.2 Safety precautions

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

The following safety precautions refer exclusively to the control as an individual component but not to the specific complete product, i.e. the machine tool.

> ⚙ Refer to your machine manual.
>
> Before using the machine including the control, take the OEM documentation to inform yourself about the safety-related aspects, the necessary safety equipment as well as the requirements on the qualified personnel.

The following overview contains exclusively the generally valid safety precautions. Pay attention to additional safety precautions that may vary with the configuration and are given in the following chapters.

> ℹ For ensuring maximum safety, all safety precautions are repeated at the relevant places within the chapters.

⚠ DANGER

Caution: hazard to the user!

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

---

⚠ DANGER

Caution: hazard to the user!

Machines and machine components always pose 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

⚠ WARNING

Caution: hazard to the user!

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

---

NOTICE

Danger of collision!

Failure to notice deviations between the actual axis positions and those expected by the control (saved at shutdown) can lead to undesirable and unexpected axis movements. There is risk of collision during the reference run of further axes 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 manufacturer

---

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

---

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

NOTICE

Danger of collision!

The control uses the defined tool length from the tool table for compensating for the tool length. Incorrect tool lengths will result in an incorrect tool length compensation. The control does not perform tool length compensation or a collision check for tools with a length of **0** and after a **TOOL CALL 0**. There is a risk of collision during subsequent tool positioning movements!

▶ Always define the actual tool length of a tool (not just the difference)
▶ Use **TOOL CALL 0** only to empty the spindle

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. If the previous value is kept, there is a danger of collision!

- ▶ Before activating a preset, check whether all columns contain values.
- ▶ For undefined columns, enter values (e.g., **0**)
- ▶ As an alternative, have the machine manufacturer define **0** as the default value for the columns

---

NOTICE

Danger of collision!

If you select an NC block in program run using the **GOTO** function and then execute the NC program, the control ignores all previously programmed NC functions (e.g., transformations). This means that there is a risk of collision during subsequent traversing movements!

- ▶ Use **GOTO** only when programming and testing NC programs
- ▶ Only use **Block scan** when executing NC programs

NOTICE

Danger of collision!

NC programs that were created on 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 **Program Run** Single Block mode

---

NOTICE

Caution: Data may be lost!

If you do not properly remove a connected USB device during a data transfer, then data may be damaged or deleted!

- ▶ Use the USB port only for transferring or backing up data do not use it for editing and executing NC programs
- ▶ Use the **Eject** soft key to remove a USB device when data transfer is complete

---

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 regardless of the control's status!

- ▶ Always shut down the control
- ▶ Only operate the main switch after being prompted on the screen

2.3　Software

This User's Manual describes the functions for setting up the machine as well as for programming and running your NC programs. These functions are available for a control featuring the full range of functions.

> **i**　The actual range of functions depends, among other things, on the enabled software options.
> **Further information:** "Software options", Page 49

The table shows the NC software numbers described in this User's Manual.

> **i**　HEIDENHAIN has simplified the version schema, starting with NC software version 16:
> - The publication period determines the version number.
> - All control models of a publication period have the same version number.
> - The version number of the programming stations corresponds to the version number of the NC software.

NC software number	Product
817620-20	TNC7 go
817625-20	TNC7 go Programming Station

> **⚙**　Refer to your machine manual.
> This User's Manual describes the basic functions of the control. The machine manufacturer can adapt, enhance or restrict the control functions to the machine. The machine manufacturer can also change the colors in the control's GUI.
>
> Check, on the basis of the machine tool manual, whether the machine manufacturer has adapted the functions of the control.
>
> If later customization of the machine configuration by the machine manufacturer is intended, the machine operator might incur additional costs.

2.3.1 Software options

Software options define the range of functions of the control. The optional functions are either machine- or application-specific. The software options give you the possibility of adapting the control to your individual needs.

You can check which software options are enabled on your machine.

Further information: "Viewing of software options", Page 338

The TNC7 go features various software options that the machine manufacturer may enable separately, even at a later point in time. The following overview includes only those software options that are relevant for you.

The software options are saved on the **SIK** (System Identification Key) plug-in board. The TNC7 go is equipped with a **SIK2** plug-in board, which means that the numbers of the software options differ from older controls, for example.

> ℹ️ The option numbers in parentheses given in the User's Manual show you that a function is not included in the standard range of available functions.
>
> The parentheses enclose the **SIK** and **SIK2** option numbers, separated by a slash, for example: (#18 / #3-03-1).
>
> The Technical Manual contains information about additional software options that are relevant to the machine manufacturer.

SIK2 definitions

SIK2 option numbers are structured by <class>-<option>-<version>:

Class	The function is effective for the following areas:
	■ 1: Programming, simulation, and process setup
	■ 2: Part quality and productivity
	■ 3: Interfaces
	■ 4: Technology functions and quality assessment
	■ 5: Process stability and monitoring
	■ 6: Machine configuration
	■ 7: Developer tools
Option	Sequential number within each class
Version	New versions of software options are released if, for example, its features have been changed.

You can order some software options with **SIK2** more than once in order to obtain multiple variants of the same function (e.g., if you need to enable multiple control loops for the axes). In the User's Manual, these software option numbers are identified by an asterisk (*****).

The control indicates in the **SIK** menu item of the **Settings** application whether a software option has been enabled, and if so, how often. The control also shows whether is equipped with the **SIK** or **SIK2**.

Further information: "The SIK menu item", Page 337

Overview

> ℹ️ Keep in mind that particular software options also require hardware extensions.
> **Further information:** "Hardware", Page 52

Software option	Definition and application
Control Loop Qty. (#0-3 / #6-01-1*)	**Additional control loop** A control loop is required for each axis or spindle moved to a programmed nominal value by the control. Additional control loops are required, for example, for detachable and motor-driven tilting tables. If your control features a **SIK2**, you can order this software option multiple times and enable up to control loops.
Touch Probe Functions (#17 / #1-05-1)	**Touch-probe functions** This software option is used to program and execute automatic probing processes. If you are using a HEIDENHAIN touch probe with EnDat interface, then the Touch Probe Functions software option (#17 / #1-05-1) is automatically enabled. The software option includes the following functions: ■ Automatic compensation of workpiece misalignment ■ Automatic setting of workpiece presets ■ Automatic measurement of workpieces ■ Automatic measurement of tools The touch-probe functions reduce the setup effort and increase accuracy when machining workpieces. **Further information:** "Touch probe functions in the Manual operating mode (#17 / #1-05-1)", Page 209
HEIDENHAIN DNC (#18 / #3-03-1)	**HEIDENHAIN DNC** This software option enables external Windows applications to access data of the control via the TCP/IP protocol. Potential fields of application include: ■ Connection to higher-level ERP or MES systems ■ Capture of machine and operating data HEIDENHAIN DNC is required in conjunction with external Windows applications.
OPC UA NC Server (#56-61 / #3-02-1*)	**OPC UA NC Server** These software options include OPC UA, a standardized interface for remote access to the control's data and functions. Potential fields of application include: ■ Connection to higher-level ERP or MES systems ■ Capture of machine and operating data Each software option enables one client connection. If more than one parallel connection is required, you need to enable multiple of these software options. If your control features the **SIK2**, you can order this software option multiple times and enable up to ten connections. **Further information:** "OPC UA NC Server (#56-61 / #3-02-1*)", Page 362

Software option	Definition and application
CAD Model Optimizer (#152 / #1-04-1)	**Optimization of CAD models** This software option can be used, for example, to repair faulty files of fixtures and tool holders or to position STL files generated from the simulation for a different machining operation. **Further information:** "Generating STL files with 3D mesh (#152 / #1-04-1)", Page 193
PTI 8 channels (#3-04-1*)	**Process Tracking Interface PTI** This software option provides an interface for recording control and process signals at a scanning interval as short as 3 ms. With PTI, you can record live information about axes and spindles. The signals can be used, among other things, to create a digital workpiece twin in order to meet documentation requirements in the aerospace industry. This software option is available only for controls with **SIK2**. You can order this software option up to two times, each time selecting up to eight channels for recording. **Further information:** "Overview of signals for Process Tracking Interface (PTI) (#3-04-1*)", Page 460

> 📖 For more information, refer to the integrated help system of RemoTools SDK.
> ID 340442-xx

2.3.2 Information on licensing and use

Open-source software

The control software contains open-source software whose use is subject to explicit licensing terms. These special terms of use have priority.

To get to the licensing terms on the control:

⌂ ▶ Select the **Home** operating mode

⚙ ▶ Select the **Settings** application

▶ Select the **Operating system** tab

🆗 ▶ Double-tap or double-click **About HeROS**

> The control opens the **HEROS License Viewer** window.

OPC UA

The control software contains binary libraries, to which the terms of use agreed between HEIDENHAIN and Softing Industrial Automation GmbH additionally and preferentially apply.

The control's behavior can be influenced by means of the OPC UA NC Server (#56-61 / #3-02-1*) and HEIDENHAIN DNC (#18 / #3-03-1). Before using these interfaces for productive purposes, system tests must be performed to exclude the occurrence of any malfunctions or performance failures of the control. The manufacturer of the software product that uses these communication interfaces is responsible for performing these tests.

Further information: "OPC UA NC Server (#56-61 / #3-02-1*)", Page 362

2.4 Hardware

This User's Manual describes functions for setting up and operating the machine. These functions primarily depend on the installed software.

Further information: "Software", Page 48

The actual range of functions also depends on hardware enhancements and the enabled software options.

Further information: "Hardware enhancements", Page 56

Further information: "Software options", Page 49

The control must have at least 16 GB of RAM, as the control will otherwise display a warning.

2.4.1 Touchscreen and keyboard unit
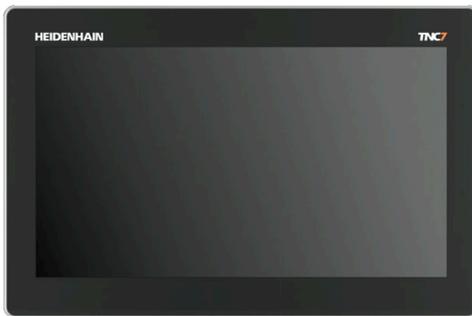
16" MC 345

The TNC7 go is delivered with a 16-inch screen.

The control is operated by means of touchscreen gestures and with the controls of the keyboard unit.

Further information: "Common gestures for the touchscreen", Page 65

Further information: "Operating elements of the keyboard unit", Page 65

The machine operating panel is machine-dependent.

MB 340 (FS)

Operating and cleaning the touchscreen

Touchscreens can even be operated with dirty hands, as long as the touch sensors are able to detect the skin resistance. Small amounts of liquid do not affect the function of the touchscreen, but large amounts may cause incorrect input.

Switch off the control before cleaning the touchscreen. As an alternative, you can use the touchscreen cleaning mode.

To navigate to this function:

Settings application ▶ Diagnostics/Maintenance group ▶ Touchscreen Cleaning

Further information: "The Settings application", Page 327

Do not apply the cleaning agent directly to the screen, but slightly dampen a clean, lint-free cleaning cloth with it.

The following cleaning agents are permitted for the screen:

- Glass cleaner
- Foaming screen cleaners
- Mild detergents

The following cleaning agents are prohibited for the screen:

- Aggressive solvents
- Abrasives
- Compressed air
- Steam cleaners

ⓘ
- Touchscreens are sensitive to electrostatic charges from the user. Dissipate the static charge by touching metallic, grounded objects or wear ESD clothing.
- Wear operating gloves to prevent the screen from getting dirty.
- You can operate the touchscreen with special touchscreen operating gloves.

Cleaning the keyboard unit

Switch the control off before cleaning the keyboard unit.

NOTICE

Caution: risk of property damage

Incorrect cleaning agents and incorrect cleaning procedures can damage the keyboard unit or parts of it.

▶ Use permitted cleaning agents only

▶ Use a clean, lint-free cleaning cloth to apply the cleaning agent

---

The following cleaners are permitted for the keyboard unit:

- Cleaning agents containing anionic surfactants
- Cleaning agents containing nonionic surfactants

The following cleaning agents are prohibited for the keyboard unit:

- Cleaning agents for machines
- Acetone
- Aggressive solvents
- Abrasives
- Compressed air
- Steam cleaners

> 🛈 Wear operating gloves to prevent the keyboard unit from getting dirty.

If a trackball is embedded in the keyboard, you need to clean it only if it no longer works properly.

To clean a trackball (if needed):

▶ Shut down the control

▶ Turn the pull-off ring by 100° in counterclockwise direction

> Turning the removable pull-off ring moves it upwards out of the keyboard unit.

▶ Remove the pull-off ring

▶ Take out the ball

▶ Carefully remove sand, chips, or dust from the shell area

> 🛈 Scratches in the shell area may impair the functionality or prevent proper functioning.

▶ Apply a small amount of the cleaning agent onto a cleaning cloth

▶ Carefully wipe the shell area clean with the cloth until all smears or stains have been removed

Exchanging keycaps

If you need replacements for the keycaps of the keyboard unit, contact HEIDENHAIN or the machine manufacturer.

> ⚙ IP54 protection cannot be guaranteed if the keyboard is missing any keys.

To exchange the keycaps:
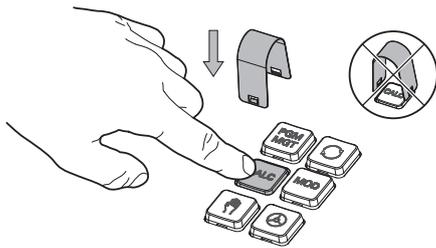
▶ Slide the keycap puller (ID 1325134-01) over the keycap until the grippers engage

> 🛈 Pressing the key will make it easier to apply the keycap puller.
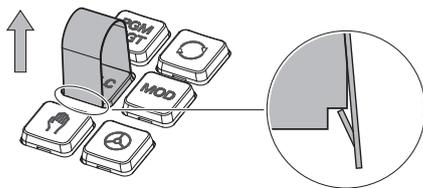
▶ Pull off the keycap
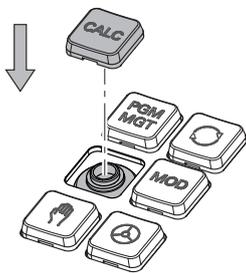
▶ Place the keycap onto the seal and push it down

> 🛈 The seal must not be damaged; otherwise IP54 protection cannot be guaranteed.

▶ Verify proper seating and correct functionality

2.4.2 Hardware enhancements

The hardware enhancements give you the possibility of adapting the machine tool to your individual needs.

The TNC7 go features various hardware extensions that the machine manufacturer may add separately, even at a later point in time. The following overview includes only those extensions that are relevant to you.

> **i** Keep in mind that particular hardware enhancements require additional software options.
>
> **Further information:** "Software options", Page 49

Hardware enhancements	Definition and application
Electronic handwheels	You use this enhancement for exact manual positioning of machine axes. The wireless portable variants improve ergonomics and increase versatility.
	The handwheels have the following differing features:
	■ Portable or installed in the machine operating panel
	■ With or without display
	■ With or without functional safety
	Electronic handwheels, for example, greatly simplify workpiece setup.
	Further information: "Electronic handwheel", Page 301
Workpiece touch probes	The control uses this enhancement for automatic and precise detection of workpiece positions (#17 / #1-05-1).
	The workpiece touch probes have the following differing features:
	■ With radio or infrared transmission
	■ With or without cable
	Workpiece touch probes, for example, are useful for quick workpiece setup and for automatic correction of dimensions during program run.
	Further information: "Touch probe functions in the Manual operating mode (#17 / #1-05-1)", Page 209
Tool touch probes	With this extension, the control can measure tools automatically and precisely, directly in the machine (#17 / #1-05-1).
	Tool touch probes have the following differing features:
	■ Contact-free or tactile measurement
	■ With radio or infrared transmission
	■ With or without cable
	Tool touch probes, for example, are useful for quick workpiece setup and for automatic correction of dimensions and breakage control during program run.
	Further information: Programming and Testing User's Manual
Industrial PC	You use this enhancement to install and run Windows-based applications.
	The industrial PC is a secure and powerful alternative to external PCs.

2.5 Areas of the control's user interface
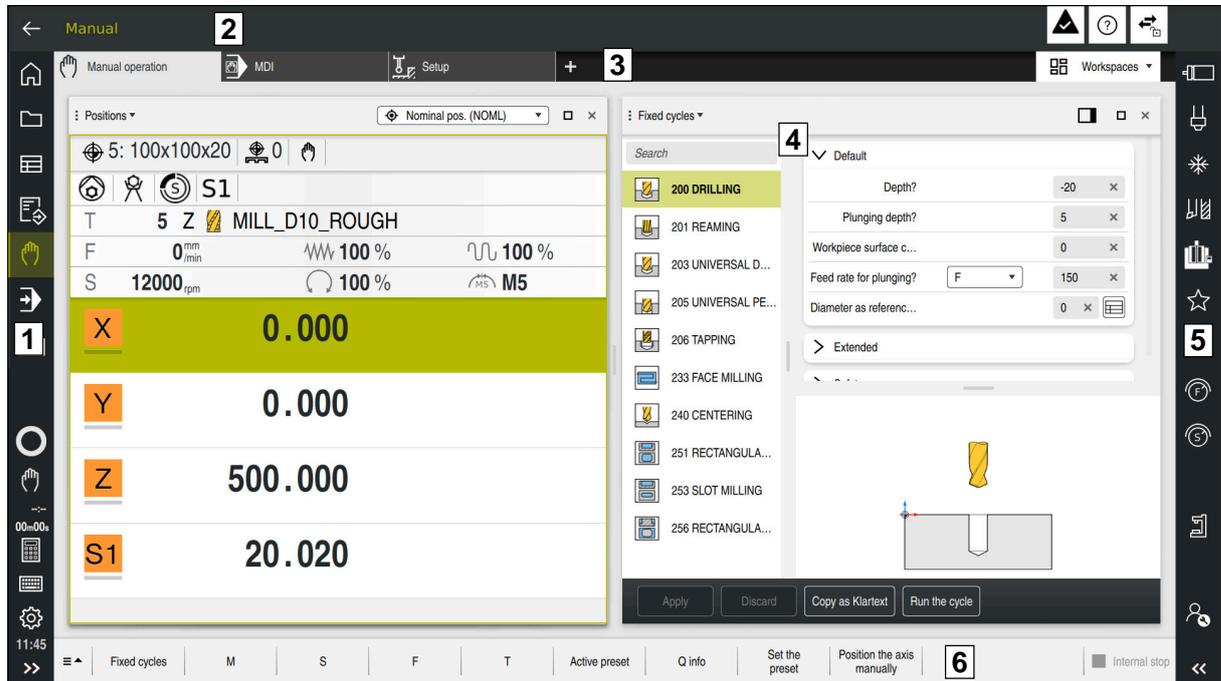
User interface of the control in the **Manual operation** application

The control's user interface shows the following areas:

1 TNC bar
- Back

  Use this function to go backwards in the application history since booting the control.
- Operating modes

  Further information: "Overview of the operating modes", Page 59
- Status overview

  Further information: "Status overview of the TNC bar", Page 99
- Calculator

  Further information: Programming and Testing User's Manual
- Screen keyboard

  Further information: "Virtual keyboard of the control bar", Page 198
- Settings

  The Settings menu enables you to change the control interface:
  - **Left-hand mode**

    The control swaps the positions of the TNC bar and the machine manufacturer bar.
  - **Dark Mode**

    In the machine parameter **darkModeEnable** (no. 135501), the machine manufacturer defines whether the **Dark Mode** function is available for selection.

    This manual describes the state of the GUI with inactive **Dark Mode**. Your control might display other colors than shown in this User's Manual.
  - **Font size**

- Date and time
2 Information bar
    - Active operating mode
    - Message menu
      Further information: "Message menu on the information bar", Page 201
    - Symbols
      Further information: "Icons on the control's user interface", Page 73
3 Application bar
    - Tabs of opened applications
      The maximum number of simultaneously opened applications is limited to ten tabs. If you try to open an eleventh tab, the control shows a message.
    - Selection menu for workspaces
      With the selection menu you define which workspaces are open in the active application.
4 Workspaces
    Further information: "Workspaces", Page 61
5 Machine manufacturer bar
    The machine manufacturer configures the machine manufacturer bar.
6 Function bar
    - Selection menu for buttons
      With the selection menu you define which buttons the control displays in the function bar.
    - Button
      With the buttons you activate individual functions of the control.

> Refer to your machine manual.
> The machine parameter **state** (no. 143601) allows the machine manufacturer to dim or hide the following user interface elements, for example:
> - Operating modes
> - Applications
> - Workspaces
> - Buttons

2.6 Overview of the operating modes

The control provides the following operating modes:

Icon	Operating modes	Further information
⌂	The **Home** operating mode contains the following applications: ■ The **Start/Login** application During the startup process, the control is in the **Start/Login** application.	
	■ The **Settings** application	Page 327
	■ The **Help** application	See the User's Manual for Programming and Testing
	■ Applications for machine parameters	Page 391
▢	In the **Files** operating mode the control displays drives, folders and files. You can, for example, create or delete folders or files and can also connect drives.	See the User's Manual for Programming and Testing
▦	In the **Tables** operating mode you can open various tables and edit them as necessary.	Page 256
✎	In the **Editor** operating mode you can do the following: ■ Create, edit and simulate NC programs	See the User's Manual for Programming and Testing
✋	The **Manual** operating mode contains the following applications:	
	■ The **Manual operation** application	Page 124
	■ The **MDI** Application	Page 205
	■ The **Setup** application	Page 209
	■ The **Move to ref. point** application	Page 120
	■ The **Retract** application You can move the tool away from the workpiece, for example after a power failure.	Page 252
⇥	In the **Program Run** operating mode you produce workpieces by having the control execute NC programs either block-by-block or in full sequence.	Page 230
🖳	In the **Machine** operating mode, the machine manufacturers define their own functions, such as diagnostic functions for spindle and axes, or other applications. Refer to your machine manual.	

⚙️ Refer to your machine manual.

The machine parameter **state** (no. 143601) allows the machine manufacturer to gray out or hide the following user interface elements, for example:

- Operating modes
- Applications
- Buttons

2.7 Workspaces

2.7.1 Operating elements within the workspaces
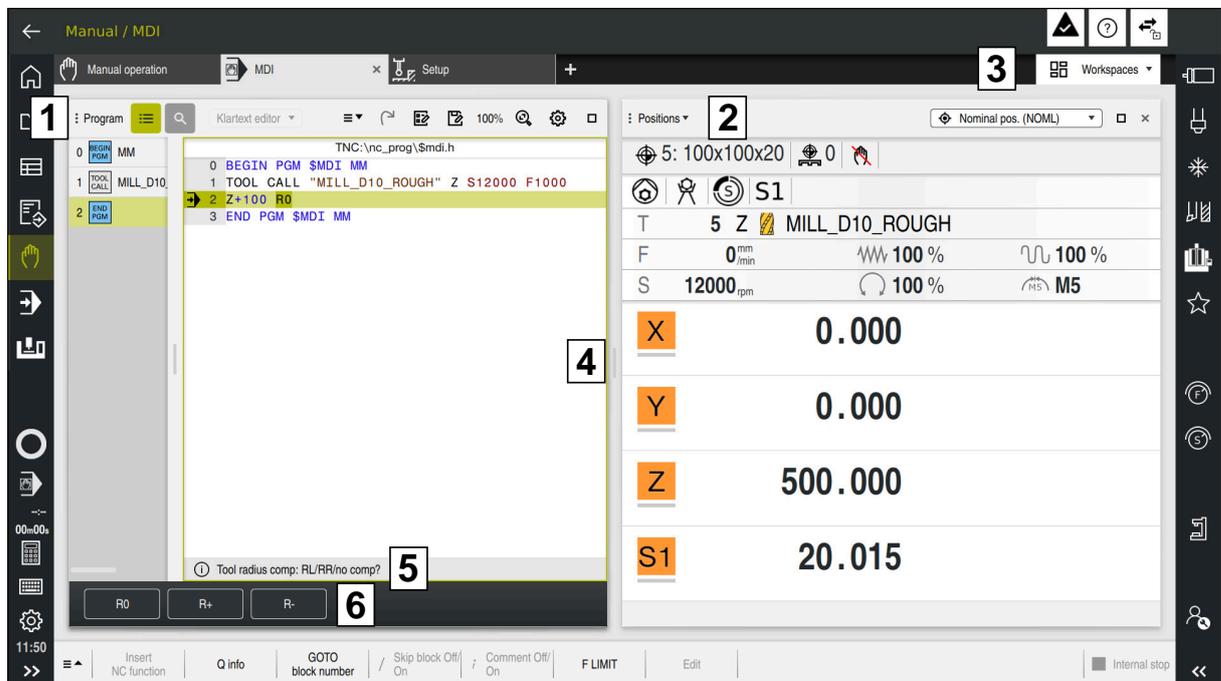
The control in the **MDI** application with two open workspaces.

The control displays the following operating elements:

1 The **Move** icon

Use the **Move** icon in the title bar to change the position of the workspaces. You can also align two workspaces vertically above each other.

Alternatively, you can drag the title to move a workspace.

2 Title bar

In the title bar the control shows the title of the workspace, and different icons or settings, depending on the workspace.

When you select the title of a workspace, the control will open a selection menu. This selection menu allows you to switch workspaces without changing their size or position.

You can only switch between workspaces that can be closed.

3 Selection menu for workspaces

Use the selection menu for workspaces in the application bar to open individual workspaces. The available workspaces depend on the active application.

4 Separator

You use the separator between two workspaces to change the scaling of the workspaces.

5 Dialog bar

In the dialog bar, the control shows information on the current input, such as a description or unit of measure.

6 Action bar

In the action bar the control shows selection options for the current dialog; for example, an NC function.

2.7.2 Icons within the workspaces

In the title bar, the control displays the following icons:

Icon	Meaning
▼	**Open the Workspaces selection menu** Only for workspaces that can be closed
◻	**Maximize** Only if at least two workspaces are open
⊡	**Reduce** Only if at least two workspaces are open
✕	**Close** Only if at least two workspaces are open that can be closed

If you maximize a workspace, the control shows the workspace over the application's entire area. If you reduce the workspace, then all other workspaces return to their previous position.

Icons that are specific to individual workspaces are described there.

An overview of the icons that are accessible from all operating modes or can be used in multiple operating modes is found here:

Further information: "Overview of icons not specific to any operating mode", Page 73

2.7.3 Overview of workspaces

The control offers the following workspaces:

Workspace	Further information
Probing function (#17 / #1-05-1) In the **Probing function** workspace you set presets on the workpiece and . You can also calibrate the touch probe, measure tools.	Page 209
Fixed cycles In the **Fixed cycles** workspace, you can define a cycle the control will execute at the current position. The machine manufacturer defines the available cycles.	Page 132
Open File In the **Open File** workspace you select or create files, for example.	See the User's Manual for Programming and Testing
Files In the file manager, the control displays drives, folders, and files. You can, for example, create or delete folders or files and can also connect drives. The **Files** workspace is part of the **Files** operating mode.	See the User's Manual for Programming and Testing
Details In the **Details** workspace, the control displays information on the selected machine parameter or the last change you made.	Page 396
Document You can open files for viewing in the **Document** workspace, for example a technical drawing.	See the User's Manual for Programming and Testing

Workspace	Further information
Settings	Page 327
In the **Settings** workspace, you can display and edit, if required, various settings of the control (e.g., set up the traverse limits).	
The **Settings** workspace is part of the **Settings** application.	
The **Form** for tables	Page 270
In the **Form** workspace, the control shows all contents of a selected table row. Depending on the table, you can edit the values in the form.	
Retract	Page 252
In the **Retract** workspace, you can disengage the tool after a power interruption.	
Graphic Positioning	Page 135
In the **Graphic Positioning** workspace, you can define a cycle the control will execute at multiple positions. You can define the machining positions graphically.	
Desktop menu	Page 76
In the **Desktop menu** workspace, the control displays selected control and HEROS functions.	
Help	See the User's Manual for Programming and Testing
In the **Help** workspace, the control displays a help graphic for the current syntax element of an NC function or the integrated product aid **TNCguide**.	
List	Page 392
In the **List** workspace, the control shows the machine parameter structure; you might be able to edit some of the parameters.	
Positions	Page 95
In the **Positions** workspace, the control displays information about the status of various functions of the control and about current axis positions.	
Program	See the User's Manual for Programming and Testing
The control displays the NC program in the **Program** workspace.	
Referencing	Page 120
On machines with incremental linear and angle encoders, the control shows in the **Referencing** workspace which axes need to be referenced.	
Quick selection	See the User's Manual for Programming and Testing
In the **Quick selection new table** and **Quick selection new file** workspaces, you can create files or open existing files, depending on the active operating mode.	
Simulation	See the User's Manual for Programming and Testing
In the **Simulation** workspace, the control shows the simulated or current movements, depending on the operating mode.	
Simulation status	Page 110
In the **Simulation status** workspace the control shows data based on the simulation of the NC program.	
Start	Page 81
In the **Start** workspace, the control shows the steps that are performed during startup.	

Workspace	Further information
Status In the **Status** workspace, the control shows the status and values of individual functions.	Page 101
Table In the **Table** workspace, the control shows the contents of a table. You can search in all tables and filter the table content.	Page 262
The **Table** for machine parameters In the **Table** workspace the control shows the machine parameters; you might be able to edit some of them.	Page 392
Keyboard In the **Keyboard** workspace, you can enter NC functions, letters and numbers, and also navigate.	Page 198
Text editor Use the **Text editor** workspace to create and edit text files.	See the User's Manual for Programming and Testing
Overview In the **Overview** workspace, the control displays information on the status of individual functional safety (FS) safety functions.	Page 322

⚙ Refer to your machine manual.

The machine parameter **state** (no. 143601) allows the machine manufacturer to dim or hide the following user interface elements, for example:

- Operating modes
- Applications
- Workspaces
- Buttons

2.8 Operating elements

2.8.1 Common gestures for the touchscreen

The screen of the control is multi-touch capable. That means the control can distinguish various gestures, even with two or more fingers at once.

You can use the following gestures:

Icon	Gesture	Meaning
●	Tap	Select element
◉	Double tap	■ Open an element (e.g., window in the **Settings** application) ■ Edit an NC block ■ Reset the graphic or 3D model to its original size
●	Long press	Open context menu ⓘ ■ If you are working with a mouse, click with the right mouse key. ■ If you do not stop holding, the control will automatically cancel the holding gesture after approximately ten seconds.
●	Swipe	■ Scroll ■ Rotate the graphic or 3D model
●	Drag	■ Change the selected area ■ Shift elements
●●	Two-finger drag	■ Move a graphic or 3D model ■ Shift drawing view in the **Contour graphics** workspace
●●	Spread	■ Change font size ■ Enlarge a graphic or 3D model
●●	Pinch	■ Reduce font size ■ Reduce a graphic or 3D model

2.8.2 Operating elements of the keyboard unit

Application

You operate the TNC7 go primarily through the touchscreen, meaning with gestures.

Further information: "Common gestures for the touchscreen", Page 65

In addition, the control's keyboard unit offers keys and other elements for alternative operating sequences.

Description of function

The tables below describe the keyboard unit's operating elements.

> **i** If there are deviations from the virtual keyboard, the table also indicates the corresponding keys on the virtual keyboard.
>
> **Further information:** "Virtual keyboard of the control bar", Page 198

Keycaps for alphabetic keyboard

Key	Meaning
A B C	Enter texts (e.g., file names)
Q Q QL QR	**Q** With an open NC program, enter a Q parameter formula in the **Editor** operating mode, or in the **Manual** operating mode open the **Q parameter list** window **Further information:** Programming and Testing User's Manual By selecting the **Q** key multiple times, you can switch between **Q**, **QL**, and **QR**.
ESC	Close the window or context menu, discard the current input
⇄	Select the next element; for example, an input field, button, or selection option
[SHIFT] + [TAB]	Select the previous element
⬸	Show the hidden NC block
? /	Hide or show NC blocks
PRT SC	Create screenshot
⊘	The **DIADUR** keys provide the following functions: ■ Left **DIADUR** key Open the **HEROS menu**
▤	Open context menu
+ = − − * 8 ? / (9) 0	Perform calculations for numerical values in input fields and table cells

Keycaps for operating aids

Key	Meaning
PGM MGT	Open the **Open File** workspace in the **Editor** and **Program Run** operating modes **Further information:** Programming and Testing User's Manual
⭕	Currently no function
ERR	Open and close the message menu **Further information:** "Message menu on the information bar", Page 201
CALC	Open and close the calculator **Further information:** Programming and Testing User's Manual
MOD	Open the **Settings** application **Further information:** "The Settings application", Page 327
HELP	Open the online help **Further information:** "User's Manual as integrated product aid: TNCguide", Page 35

Operating modes

> ℹ️ On the TNC7 go, the operating modes of the control are allocated differently than on the previous controls. For reasons of compatibility and to facilitate ease of operation, the keys on the keyboard unit remain the same. Keep in mind that particular keys no longer activate a change of operating modes but, for example, instead activate a toggle switch.

Key	Meaning
🖐️	Open the **Manual operation** application in the **Manual** operating mode **Further information:** "The Manual operation application", Page 124
⊕	Activate and deactivate the electronic handwheel in the **Manual** operating mode **Further information:** "Electronic handwheel", Page 301
⊒▶	Open the **Tool Management** tab in the **Tables** operating mode **Further information:** "Tool management ", Page 163
▣▶	Open the **MDI** application in the **Manual** operating mode **Further information:** "The MDI application ", Page 205
⊟▶	Open the **Program Run** operating mode in **Single Block** mode **Further information:** "The Program Run operating mode", Page 230
⊐▶	Open the **Program Run** operating mode **Further information:** "The Program Run operating mode", Page 230
⬦	Open the **Editor** operating mode **Further information:** Programming and Testing User's Manual
⊐▶	While the NC program is running, open the **Simulation** workspace in the **Editor** operating mode **Further information:** Programming and Testing User's Manual

Keycaps for NC dialog

> **i** The following functions are valid for the **Editor** operating mode and the **MDI** application.

Key	Meaning
TOUCH PROBE	In the **Insert NC function** window, open the **Setup** folder in order to select a touch probe cycle **Further information:** Programming and Testing User's Manual
CYCL DEF	In the **Insert NC function** window, open the **Fixed cycles** folder in order to select a cycle **Further information:** Programming and Testing User's Manual
CYCL CALL	In the **Insert NC function** window, open the **Cycle call** folder to select a machining cycle. **Further information:** Programming and Testing User's Manual
LBL SET	Program a jump label
LBL CALL	Program a subprogram or a program section repeat
STOP	Program an intentional stop
TOOL DEF	Pre-select a tool in the NC program
TOOL CALL	Call the tool in the NC program
SPEC FCT	In the **Insert NC function** window, open the **Special functions** folder (e.g., to program a workpiece blank)
PGM CALL	In the **Insert NC function** window, open the **Selection** folder (e.g., to call an external NC program)

Further information: Programming and Testing User's Manual

Keycaps for axis input and value input

Key		Meaning
X	V	Select axes in the **Manual** operating mode, or enter them in the **Editor** operating mode
0	9	Enter numbers (e.g., coordinate values)
.		Insert a decimal separator during entry
−/+		Invert algebraic sign of entered value
⟨X⟩		Delete values during entry
✛		Open position display of the status overview to copy axis values **Further information:** "Status overview of the TNC bar", Page 99 In the **Editor** operating mode and the **MDI** application, take over the actual position of the defined axis
Q FN		In the **Editor** operating mode, open the **FN** folder in the **Insert NC function** window
CE		Clear entries or delete messages
DEL		Delete NC block or cancel a dialog during programming
NO ENT		Skip or remove optional syntax elements during programming
ENT		Confirm entries and continue dialogs
END		Conclude entry (e.g., finish an NC block)
I		Switch between entry of incremental and absolute coordinates

Keycaps for navigation

Key	Meaning
↑ ... → ▲ ... ▶	Position the cursor
GOTO	■ Directly position the cursor by using the number of an NC block, table row or machine parameter ■ Open the selection menu while editing
HOME	Jump to first line of an NC program or first column of a table
END	Jump to last line of an NC program or last column of a table
PG UP	Go one page up in an NC program or table
PG DN	Go one page down in an NC program or table
	Mark the active application in order to navigate between applications
	Navigate between areas of an application

Potentiometers

Potentiometer	Function
100 50 150 0 F %	Increase or reduce the feed rate **Further information:** Programming and Testing User's Manual
100 50 150 0 S %	Increase or reduce the spindle speed **Further information:** Programming and Testing User's Manual
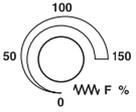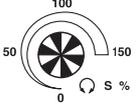
2.8.3   Keyboard shortcuts for operating the control

With a keyboard unit or a USB keyboard, you can use keyboard shortcuts in your control. In the User's Manual, the labels of the keys are used when indicating keyboard shortcuts. Keys without a label are indicated as follows:

Key	Designation
⇧	SHIFT
	SPACE
↵	RETURN
⇄	TAB
↑	UP
↓	DOWN
→	RIGHT
←	LEFT

2.8.4 Icons on the control's user interface

Overview of icons not specific to any operating mode

This overview describes icons that are used in more than one operating mode or that are available regardless of operating mode.

Icons that are specific to individual workspaces are described there.

Icon or shortcut	Meaning
←	**Back**
⌂	Select the **Home** operating mode
▭	Select the **Files** operating mode
▦	Select the **Tables** operating mode
⊟	Select the **Editor** operating mode
✋	Select the **Manual** operating mode
⇥	Select the **Program Run** operating mode
▥	Select the **Machine** operating mode
▦	Open or close **Calculator**
▦	**Keyboard** for formula input
⌨	Open or close **Screen keyboard**
⚙	Open or close the **Settings** selection menu
»	**Open or close** ■ White: expand the TNC bar or machine manufacturer's bar ■ Green: collapse the TNC bar or machine manufacturer's bar ■ Gray: Confirm message
+	**Add**
▭	**Open**
✕	**Close**
⋮	**Move** Change the position of workspaces or windows
✄	Activate or deactivate **Remember position** The control remembers the position of the window until it is shut down.
‹|›	Activate or deactivate **Change column width** You can change the width of the currently selected column.

Icon or shortcut	Meaning
⁚⁚. .⁚⁚	**Scale** Resize windows
. . .	File functions are available
⭐	▪ Black: **Add favorite** ▪ Yellow: **Remove favorite**
✎	**Edit**
💾 **[CTRL] + [S]**	**Save**
🖫✎ 	**Save as**
🔍 **[CTRL] + [F]**	**Find**
✂ **[CTRL] + [X]**	**Cut**
⧉ **[CTRL] + [C]**	**Copy**
📋 **[CTRL] + [V]**	**Paste**
↶ **[CTRL] + [Z]**	**Undo**
↷ **[CTRL] + [Y]**	**Redo**
100%	Font size in the workspace ℹ If you select the percent value, the control displays icons for increasing and decreasing the font size.
⊖ **[CTRL] + [-]**	**Reduce**
⊕ **[CTRL] + [+]**	**Magnify**
⊙	Set the workspace font size to 100%
▼	Open or close the selection menu
▲	ℹ Depending on the size of the workspace, the control gathers, for example, the icons in the title bar into a selection menu.
⊞	Open or close the **Workspaces** selection menu

Icon or shortcut	Meaning
◣	Show the **Message menu**
⑦	Call context-sensitive help **Further information:** "Context-sensitive help", Page 38
⇄🔒	**Secure connection configuration** External access to the control is active; all connections are using a secure connection configuration.
⇄🔓	**Non-secure connection configuration** External access to the control is active and at least one connection uses a non-secure connection configuration.
🕐	Automatic program start active **Further information:** "Automatic program start", Page 248
⬚	**Window manager** Select active applications in the background (e.g., HEROS functions window)
	ITC is connected and **Enabling VNC focus** is set **Further information:** "The VNC Focus Settings area", Page 378 Both the control and the ITC show the following icons.
✓	The focus is on the currently used device You can use the mouse and keyboard for the local and the remote device.
✗	The focus is on the remote device You cannot use the mouse and keyboard on the local device.
?	No focus assigned Inputs with the mouse and keyboard are locked until the focus has been assigned to the local or remote device.
Safety self-test	Self-test of the control is running
[CTRL] + [A]	Mark everything

2.8.5   The Desktop menu **workspace**

Application

In the **Desktop menu** workspace, the control displays selected control and HEROS functions.

Description of function

The title bar of the **Desktop menu** workspace includes the following functions:

■ The **Active Configuration** selection menu

Using the selection menu, you can activate a configuration of the control interface.

Further information: "Configuring the control's user interface", Page 398

■ Full-text search

Search for functions in the workspace with the full-text search.

Further information: "Adding and removing favorites", Page 77

The **Desktop menu** workspace contains the following areas:

■ **Control**

In this area you can open operating modes or applications.

Further information: "Overview of the operating modes", Page 59

Further information: "Overview of workspaces", Page 62

■ **Tools**

In this area you can open some tools from the HEROS operating system.

Further information: "HEROS operating system", Page 429

■ **Help**

In this area you can open training videos or **TNCguide**.

Further information: "User's Manual as integrated product aid: TNCguide", Page 35

■ **Favorites**

In this area you will find the favorites that you have chosen.

Further information: "Adding and removing favorites", Page 77
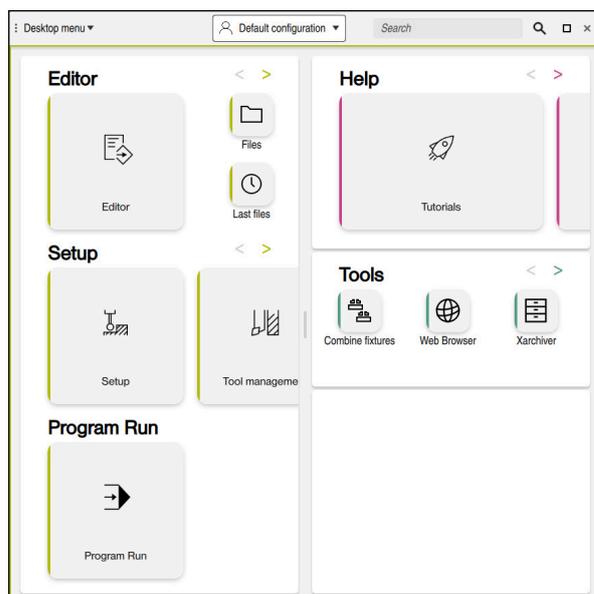
The **Desktop menu** workspace

The **Desktop menu** workspace is available in the **Start/Login** application.

Showing or hiding an area

To show or hide an area in the **Desktop menu** workspace:
▶ Hold or right-click anywhere within the workspace
> The control displays a plus sign or minus sign within each area.
▶ Select a plus sign
> The controls shows that area.

> ⓘ   Use the minus sign to hide an area.

Adding and removing favorites

Adding favorites

To add favorites in the **Desktop menu** workspace:
▶ Enter the search term in the full-text search (e.g., **dat**)
▶ Long-press or right-click the function's icon (e.g., **Date/Time**)
> The control displays the icon for **adding favorites**.

☆⁺   ▶ Select **Add favorite**
> The control adds the function to the **Favorites** area.

Removing favorites

To remove favorites from the **Desktop menu** workspace:
▶ Long-press or right-click the function's icon
> The control displays the icon for **removing favorites**.

☆⁻   ▶ Select **Remove favorite**
> The control removes the function from the **Favorites** area.

3

First steps

3.1 Chapter overview

This chapter uses an example workpiece to explain how to operate the control: from switching the machine on to the finished workpiece.

The chapter covers the following topics:

- Switching the machine and the control on
- Setting up a tool
- Setting up a workpiece
- Machining a workpiece
- Switching the machine off

3.2 Switching the machine and the control on
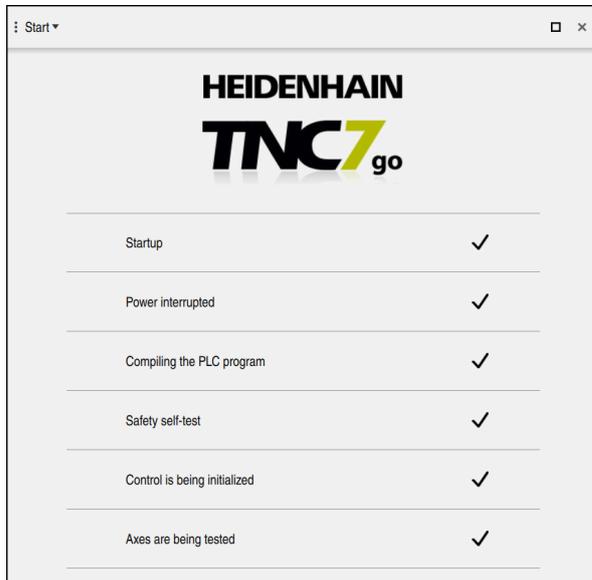
The **Start** workspace

⚠️ **DANGER**

Caution: hazard to the user!

Machines and machine components always pose 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 the machine on:

▶ Switch on the power supply of the control and of the machine
> The control is in start-up mode and shows the progress in the **Start** workspace.
> The control displays the **Power interrupted** dialog in the **Start** workspace.

OK	▶ Select **OK**
	> The control compiles the PLC program.

(I)

- ► Switch the machine control voltage on
- > The control checks the functioning of the emergency stop circuit.
- > If the machine is equipped with absolute linear and angle encoders, the control is now ready for operation.
- > If the machine is equipped with incremental linear and angle encoders, the control opens the **Move to ref. point** application.

  Further information: "The Referencing workspace", Page 120

- ► Press the **NC Start** key
- > The control moves to all necessary reference points.
- > The control is ready for operation and the **Manual operation** application is open.

  Further information: "The Manual operation application", Page 124

More detailed information

- Switching on and off
- Position encoders

  Further information: "Position encoders and reference marks", Page 147
- Axis reference run

3.3 Setting up a tool

This chapter uses an example to show how to set up a tool.

To set up a tool, the following steps are necessary:

■ Preparing the tools
■ Opening the **Tool management** application and the **Form** workspace
■ Inserting a tool in the tool management
■ Defining the tool
■ Entering the tool in the pocket table

You have to set up the tools to enable the control to position, simulate and compensate for the tools, for example.

3.3.1 Example tool

Tool	Parameter	Value
	Name	MILL_D12
	L	+120
	R	+6
	R2	+0
	TYP	**Roughing mill (MILL_R)**
	LCUTS	+20
	LU	28
	RN	4.8
	R-TIP	0
	T-ANGLE	+0
	CUTS	4
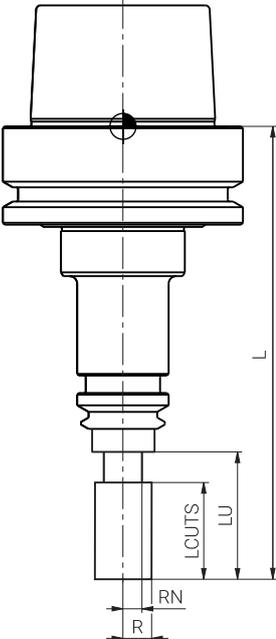
3.3.2 Preparing the tools

To prepare tools:

▶ Clamp the required tools in the tool holder
▶ Measure the tools (e.g., with a presetter)
▶ Write down the parameters
> Now the tool is ready.

ⓘ
■ You can measure tools with the tool touch probe of the control.
■ Alternatively, the control makes it possible to measure the tool without a tool touch probe.
   Further information: "Measuring the tool by scratching", Page 225
■ You cannot and do not have to measure all parameters. Some values can be found in the tool catalog for example.

3.3.3 Opening the Tool management application and the Form workspace
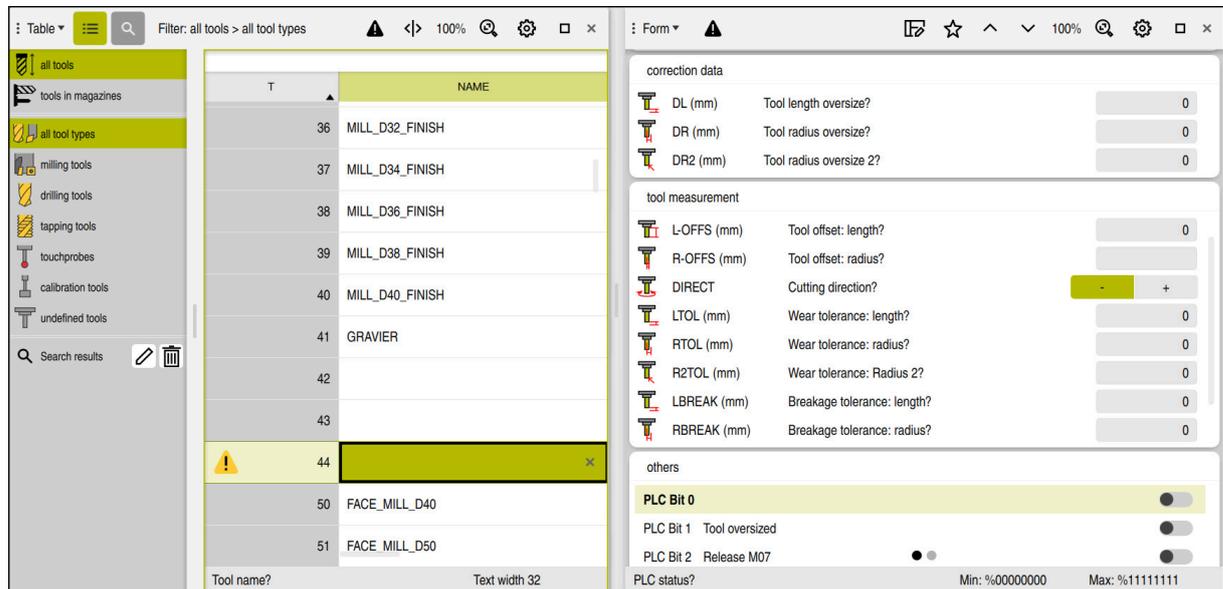
The **Tool management** application with the **Table** and **Form** workspaces

You enter tools in the **Tool management** application of the **Tables** operating mode.
You define tool parameters in the **Form** workspace. In the **Form** workspace, the control shows the required parameters of the selected tool.

To open the **Tool management** application and the **Form** workspace:

▶ Select the **Tables** operating mode
> The control displays the **Tables** operating mode.
▶ Select the **Tool management** application
> The control displays the **Tool management** application.
▶ In the application bar, select **Workspaces**
▶ Select **Form**
> The control opens the **Form** workspace.

More detailed information

▪ **Tables** operating mode
Further information: "The Tables operating mode", Page 256
▪ The **Form** workspace
Further information: "The Form workspace for tables", Page 270
▪ Tool parameters
Further information: "Tool parameters", Page 154
▪ Tool management
Further information: "Tool management ", Page 163

3.3.4 Inserting a tool in the tool management

To insert a tool:

▶ Activate **Edit**

▶ Select **Insert tool**
> The control opens the **Insert tool** window.
▶ Select the tool type, for example **Roughing mill** (**MILL_R**)
▶ Enter the line number, if required
▶ Select **OK**
> The control inserts the new tool
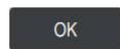
More detailed information

■ Tool types
Further information: "Tool types", Page 161

3.3.5 Defining the tool

To define the tool with the measured parameters:
▶ Enter the tool name **NAME** (e.g., **MILL_D12**)
▶ Enter the tool length **L** (e.g., **120**)
▶ Enter the tool radius **R** (e.g., **6**)
▶ Enter the tooth **LCUTS** (e.g., **20**)
▶ Enter the usable length **LU** (e.g., **28**)
▶ Enter the neck radius **RN** (e.g., **4,8**)
▶ Enter the number of teeth **CUT** (e.g., **4**)
> The control saves the tool with all entered parameters.

ⓘ Based on the entered parameters, the control shows the tool correctly in the **Simulation** workspace.

Further information: Programming and Testing User's Manual

To use the full performance range, you have to define further parameters, such as **KINEMATIC** (tool-carrier kinematics for collision monitoring). You need different parameters depending on the tool type.

Further information: "Tool parameters", Page 154

3.3.6 Entering the tool in the pocket table

> ⚙ Refer to your machine manual!
> Access to the **tool_p.tch** pocket table is machine-dependent.

The **Pocket table** application with opened **Table** workspace

The **tool_p.tch** pocket table provides the pocket assignment of the tool magazine. You must add the tool to the pocket table in order for the control to be able to insert the tool. The tool must be stored in the machine.

To enter a tool in the pocket table:

- ▶ Select the **Pocket table** application
- \> The control displays the **Pocket table** application.
- ▶ Enter the desired pocket number in the **Table** workspace
- ▶ Open the **Form** workspace
- ▶ Activate **Edit**
- ▶ Select the tool number **T**
- \> The control opens the **Tool selection** window.
- ▶ Select the tool you want to use
- ▶ Select **OK**
- \> The control inserts the tool in the pocket table.
- ▶ Define additional parameters if necessary (e.g., to reserve a pocket)
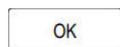
More detailed information

- Pocket table
  Further information: "Pocket table tool_p.tch", Page 290

3.4 Setting up a workpiece

3.4.1 Selecting an operating mode

You set up workpieces in the **Manual** operating mode.

To select the **Manual** operating mode:

▶ Select the **Manual** operating mode
> The control displays the **Manual** operating mode.

More detailed information

■ Operating mode: **Manual**
Further information: "Overview of the operating modes", Page 59

3.4.2 Clamping the workpiece

Mount the tool with a fixture on the machine table so that it is fixed with its edges parallel to the machine axes.

More detailed information

■ The **Probing function** workspace
Further information: "Touch probe functions in the Manual operating mode (#17 / #1-05-1)", Page 209

3.4.3 Workpiece presetting with a touch probe

Inserting a workpiece touch probe

You can set a workpiece preset with the aid of a workpiece touch probe.

To insert a workpiece touch probe:

▶ Select **T**

▶ Enter the tool number of the workpiece touch probe (e.g., **600**)

▶ Press the **NC Start** key
> The controls inserts the workpiece touch probe.

Define the workpiece preset

▶ Select the **Setup** application

▶ Select **Position (POS)**
> The control opens the **Position (POS)** touch probe function.

▶ If necessary, check the values of the active preset or choose a different preset
▶ Use the axis keys to position the workpiece touch probe at the desired probing position (e.g., above the workpiece in the workspace)

▶ Select the probing direction (e.g., **Z-**)

▶ Press the **NC Start** key
> The control performs the probing process and then automatically retracts the workpiece touch probe to the starting point.
> The control shows the measurement results.
▶ In the **Nominal value** area, enter the new preset of the probed axis (e.g., **1**)

▶ Select **Compensate the active preset**
> The control sets the active preset to the entered nominal value.

> ℹ️ Using the **Position (POS)** touch probe function, you can probe in up to three axes.

▶ Select **Exit probing**
> The control closes the **Position (POS)** touch probe function.

The **Probing function** workspace with an open manual probing function

More detailed information

■ The **Probing function** workspace
Further information: "Touch probe functions in the Manual operating mode (#17 / #1-05-1)", Page 209

■ Presets in the machine
Further information: "Presets in the machine", Page 148

■ Preset management
Further information: "Preset management", Page 179

■ Tool change in the **Manual operation** application
Further information: "The Manual operation application", Page 124

3.5 Machining a workpiece

3.5.1 Selecting an operating mode

You can machine workpieces in the **Program Run** operating mode.

To select the **Program Run** operating mode:

▶ Select the **Program Run** operating mode
> The control displays the **Program Run** operating mode and the most recently executed NC program.

More detailed information

■ The **Program Run** operating mode

Further information: "The Program Run operating mode", Page 230

3.5.2 Opening an NC program

To open an NC program:

▶ Select **Open File**
> The control displays the **Open File** workspace.

▶ Select an NC program
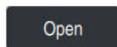
Open

▶ Select **Open**
> The control opens the NC program.

More detailed information

■ The **Open File** workspace

Further information: Programming and Testing User's Manual

3.5.3 Starting an NC program

To start an NC program:

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

3.5.4 Interrupting an NC program

To interrupt an NC program:

▶ Press the **NC stop** key
> The control will interrupt the active machining operation.

ℹ While a machining operation is interrupted, it is not possible to change the operating mode and edit the NC program.
To continue program run, press the **NC Start** key.

3.5.5 Aborting program run

After program run has been aborted, it is possible to change the operating mode and edit the NC program.

To abort program run:

▶ Press the **NC stop** key

> The control will interrupt the active machining operation.

▶ Select **Internal stop**

> The control aborts program run.

| i | For a safe restart of the NC program, always use a block scan. This ensures that the control will take the entire program contents into account. |

3.5.6 Safe restart with block scan

The **Block scan** function allows you to start an NC program at any desired NC block. The control factors workpiece machining up to this NC block into the calculations. In a block scan, the control restores the machine status up to the corresponding NC block (i.e., the spindle is switched on or the machine axes are tilted to position). A block scan reduces the risk of collision.

Running a block scan after a program interruption

To perform a block scan:

▶ Select the **Program Run** operating mode

▶ Select **Block scan**

> The control opens the **Block scan** window, displaying the data of the last executed NC block.

▶ If required, enter the **Block number** (e.g., **29**)

▶ Press the **NC Start** key

> The control starts the block scan and calculates until the entered NC block.

> The control opens the **Restore machine status** window.

▶ Press the **NC Start** key

> The control restores the machine status up to the defined NC block (e.g., **TOOL CALL** or M functions).

> The control displays the **Axis sequence for return to contour:** window. The **Axis sequence for return to contour:** window contains the positioning logic required to reach the position before the defined NC block.

▶ Press the **NC Start** key

> Using the displayed positioning logic, the control moves to the required position.

▶ Press the **NC Start** key

> The control resumes execution of the NC program.

3.6 Switching the machine off

	Refer to your machine manual. Switching off is a machine-dependent function.

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 regardless of the control's status!

▶ Always shut down the control

▶ Only operate the main switch after being prompted on the screen

---

To power-off the machine:

⌂
  ▶ Select the **Home** operating mode

Shut down
  ▶ Select **Shut down**
  > The control opens the **Shut down** window.

Shut down
  ▶ Select **Shut down**
  > If NC programs or contours contain any unsaved changes, the control displays the **Close file** window.
  ▶ If necessary, save unsaved NC programs with **Save** or **Save as**
  > The control shuts down.
  > After completion of the shutdown process, the control displays the text **Now you can switch off.**
  ▶ Switch off the main power switch of the machine

4

Status displays

4.1 Overview

The control shows the status or values of individual functions in the status displays.

The control offers the following status displays:

- General status display and position display in the **Positions** workspace
  Further information: "The Positions workspace", Page 95
- Status overview on the TNC bar
  Further information: "Status overview of the TNC bar", Page 99
- Additional status displays for specific areas in the **Status** workspace
  Further information: "The Status workspace", Page 101
- Additional status displays in the **Editor** operating mode in the **Simulation status** workspace, based on the machining status of the simulated workpiece
  Further information: "The Simulation status workspace", Page 110

4.2 The Positions workspace

Application

The general status display in the **Positions** workspace provides information about the status of various functions of the control and about current axis positions.

Description of function
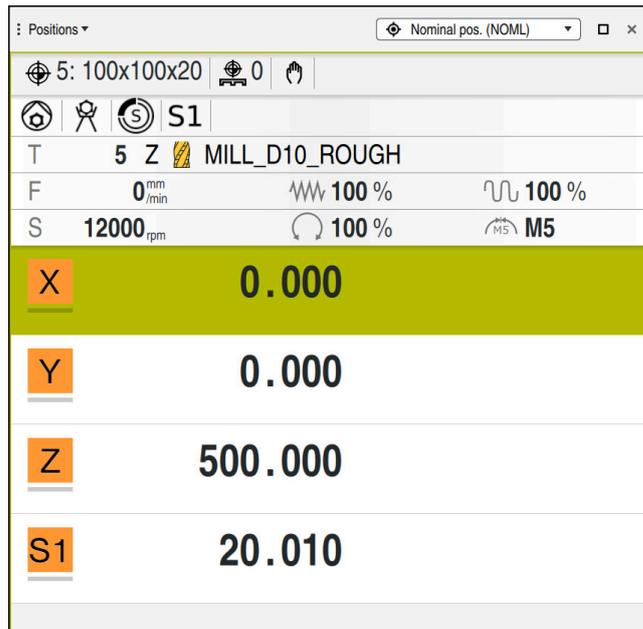
The **Positions** workspace with general status display

You can open the **Positions** workspace in the following operating modes:
- **Manual**
- **Program Run**

Further information: "Overview of the operating modes", Page 59

The **Positions** workspace provides the following information:
- Icons of active and inactive functions
- Active tool
- Technology values
- Settings of the spindle and feed-rate potentiometers
- Active miscellaneous functions for the spindle
- Axis values and statuses, such as "Axis not referenced"
  Further information: "Test status of the axes", Page 324

Axis display and position display

> ⚙ Refer to your machine manual.
>
> In the machine parameter **axisDisplay** (no. 100810) you define the quantity and sequence of the displayed axes.

Icon	Meaning
IST	Position display mode (e.g., actual or nominal coordinates of the current tool position) You can select the mode in the title bar of the workspace. **Further information:** "Position displays", Page 113
X	Axes The X axis is selected. You can move the selected axis.
m	The auxiliary axis **m** is not selected. The control displays auxiliary axes, such as the tool magazine, as lowercase letters. **Further information:** "Definition", Page 98
?	The axis is not referenced.
⚠	The axis is not in safe mode. **Further information:** "Checking axis positions manually", Page 325
Δ	The axis is moving the distance-to-go shown next to the symbol.
⊣⊢	The axis is clamped.
⊛	You can move the axis with the handwheel.
⊛🔒	You cannot move the axis with the handwheel. > ⚙ Refer to your machine manual. > The machine manufacturer defines which axes you can move with the handwheel.
F_SOS 0 mm/min	Feed status when stopped **Further information:** "Functional safety (FS) in the Positions workspace", Page 322
S_SOS 0 U/min	Spindle status when stopped **Further information:** "Functional safety (FS) in the Positions workspace", Page 322

Presets and technology values

Icon	Meaning
⊕	Number and comment of the active workpiece preset The number corresponds to the active row number of the preset table. The comment corresponds to the content of the **DOC** column. **Further information:** "Preset management", Page 179
T	In the **T** area, the control shows the following information: ◼ Number of the active tool ◼ Tool axis of the active tool ◼ Symbol of the defined tool type ◼ Name of the active tool
F	In the **F** area, the control shows the following information: ◼ Active feed rate in mm/min You can program the feed rate in various units of measurement. The control always converts the programmed feed rate in this display to mm/min. ◼ If **M136** is active: active feed rate in mm/rev **Further information:** Programming and Testing User's Manual ◼ Setting of the rapid-traverse potentiometer in percent ◼ Setting of the feed-rate potentiometer in percent **Further information:** "Potentiometers", Page 71 If a feed-rate limitation has been activated with the **F LIMIT** button, the area is labeled **F LIMIT** instead of **F**. The control displays the text **F LIMIT** and the feed-rate value in orange. **Further information:** "Feed rate limit F LIMIT", Page 234
S	In the **S** area, the control shows the following information: ◼ Active shaft speed in rpm If you have programmed a cutting speed instead of a rotational speed, the control automatically converts this value to a rotational speed. ◼ Setting of the spindle potentiometer in percent ◼ Active miscellaneous function for the spindle

Active functions

Icon	Meaning
	The **Manual traverse** function is active.
	The **Manual traverse** function is not active. **Further information:** "The Program Run operating mode", Page 230
	Tool radius compensation **R+** is active. **Further information:** Programming and Testing User's Manual
	Tool radius compensation **R-** is active. **Further information:** Programming and Testing User's Manual These symbols are transparent while the **Block scan** function of the control is active. **Further information:** "Block scan for mid-program startup", Page 240
	The **TRANS MIRROR** function or the Cycle **8 MIRRORING** is active. The axes programmed in this function or cycle are moved with mirroring. **Further information:** Programming and Testing User's Manual **Further information:** Programming and Testing User's Manual

> ⓘ In the optional machine parameter **iconPrioList** (no. 100813), you can change the sequence in which the control displays these symbols.

Definition

Auxiliary axes
Auxiliary axes are controlled by the PLC and are not included in the kinematics description. Auxiliary axes are driven, for example, hydraulically, electrically, or by an external motor. The machine manufacturer can define the tool magazine, for example, as an auxiliary axis.

4.3 Status overview of the TNC bar

Application

On the TNC bar, the control shows a status overview with the execution status, the current technology values, and the axis positions.

Description of function

General information

Status overview of the collapsed TNC bar with opened position display
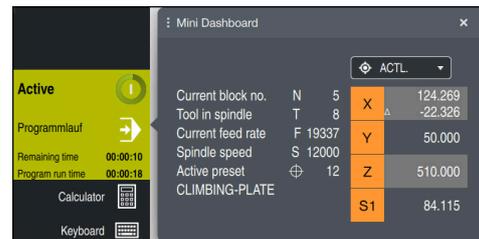
Status overview of the expanded TNC bar with opened position display

While an NC program or individual NC blocks are being executed, the control displays the following information in the status overview:

- **Control-in-operation**: Current machining status
  Further information: "Definition", Page 100
- Symbol of the application used for machining
- Remaining run time of the NC program
- Program run time
- Active tool
- Active feed rate
- Current spindle speed
- Number and comment of the active workpiece preset
- Position display

If the TNC bar is expanded, then the control displays the run times in the default format hh:mm:ss (e.g., 01:10:30).

If the TNC bar is collapsed, then the control displays the run times in the following formats:

- Minutes and seconds for run times of less than an hour (e.g., 10min30s)
- Hours and minutes for run times of one hour or longer (e.g., 01h04min)

Further information: "Display of the program run time", Page 111

Position display

If you select the status overview area, then the control opens or closes the position display with the current axis positions. The position display mode can be selected independently of the **Positions** workspace (e.g., **Actual pos. (ACT)**).

Further information: "The Positions workspace", Page 95

If you select an axis line, the control copies the current value of this line to the clipboard.

Press the **actual position capture** key to open the position display. The control prompts you to select the value to be copied to the clipboard. If you select the value of an axis while editing an NC block, then the control inserts the value into the NC block.

Further information: Programming and Testing User's Manual

Definition

Control-in-operation:

The control uses the **Control-in-operation** symbol to show the machining status of the NC program or NC block:

- White: no movement command
- Green: active machining, axes are moving
- Orange: NC program interrupted
- Red: NC program stopped

Further information: "Interrupting, stopping or canceling program run", Page 235

When the control bar is expanded, the control shows additional information about the current status, such as **Active, feed rate at zero**.

4.4 The Status workspace

Application

In the **Status** workspace the control shows the additional status display. The additional status display shows the current status of various functions on specific tabs. You can use the additional status display to better monitor the running of an NC program by receiving real-time information about active functions and accesses.

Description of function

You can open the **Status** workspace in the following operating modes:

- **Manual**
- **Program Run**

Further information: "Overview of the operating modes", Page 59

Icons

The following icons are shown in the **Status** workspace:

Icon	Meaning
	Configure the layout
	You can make the following layout adaptations:
	■ Add or remove areas to the **Favorites** view
	■ Rearrange areas using the gripper
	■ Add or remove columns
	Settings
	Some areas have their own settings. Use this icon to customize the contents of the area (e.g., by defining the variable range to be displayed).
	Favorite
	Further information: "The Favorites tab", Page 102
	Add
	The control only shows this icon when you are adapting the layout.
	With this icon you can add the following elements:
	■ Column
	You can divide the workspace into several columns.
	Further information: "Adding a column in the workspace", Page 272
	■ Area
	In the **Favorites** view you can add another area.
	Remove
	The control only shows this icon when you are adapting the layout.
	You can delete an empty column with this icon.

The Favorites tab

On the **Favorites** tab, you can arrange your own status display with the contents of the other tabs.
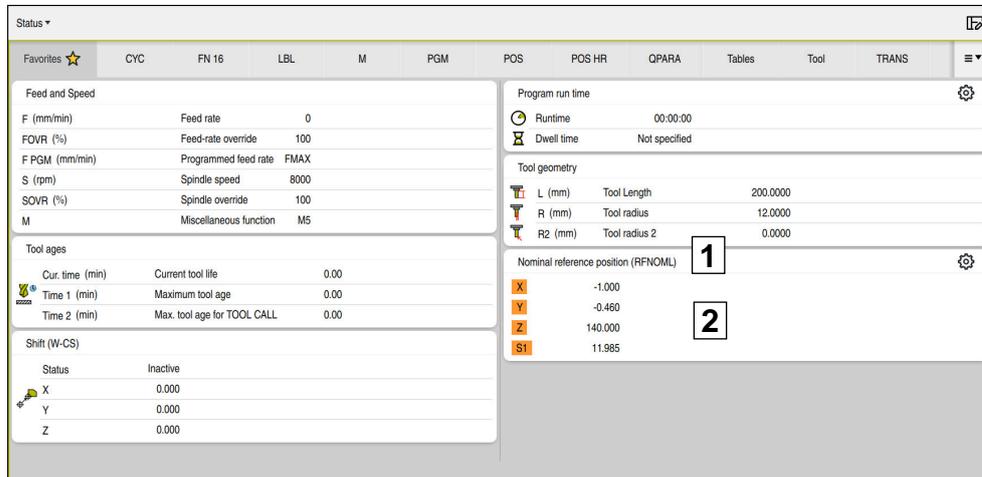
The **Favorites** tab

1   Area
2   Contents

Each section of the status display has its own **Favorites** icon. If you select the icon, the control adds that area to the **Favorites** tab.

CYC tab

On the **CYC** tab the control shows information about machining cycles.

Area	Contents
Active cycle definition	When you use the **CYCL DEF** function to define a cycle, the control shows the cycle number in this area.

The FN 16 tab

On the **FN 16** tab, the control displays the contents of a file output to the screen with **FN 16: F-PRINT**.

Further information: Programming and Testing User's Manual

Area	Contents
Output	Contents of an output file that was output with **FN 16: F-PRINT**, such as measured values or texts. To stop the output: ■ Define the **SCLR:** output path (Screen Clear) ■ Select the **Clear** button ■ Select the **Reset program** button ■ Select a new NC program

The LBL tab

On the **LBL** tab the control shows information about program section repeats and subprograms.

Further information: Programming and Testing User's Manual

Area	Contents
Subprogram calls	■ **Blk. no.** Block number of the call ■ **LBL no./Name** Called label
Repetitions	■ **Blk. no.** ■ **LBL no./Name** ■ **Program-section repeat** Number of repetitions still to be performed (e.g., 4/5)

The M tab

On the **M** tab the control shows information about active miscellaneous functions.

Further information: Programming and Testing User's Manual

Area	Contents
Active M functions	■ **Function** Active miscellaneous functions, such as **M3** ■ **Description** Descriptive text about the respective miscellaneous function. ⚙ Refer to your machine manual. Only the machine manufacturer can create a descriptive text for machine-specific miscellaneous functions.

The PGM tab

On the **PGM** tab the control shows information about the program run.

Area	Contents
Parts counter	**Quantity** Actual value and nominal value of the parts counter defined with the **FUNCTION COUNT** function **Further information:** Programming and Testing User's Manual The **Settings** icon lets you open the **Counter settings** window, in which you can check and, if needed, edit the current count and the target value for the counter. You cannot edit these values while the control is executing an NC program. ⚙ With the optional machine parameter **userPermission** (no. 129101), the machine manufacturer defines which counter settings you can change.

Program run time	■ **Runtime**
	Run time of the NC program in the format hh:mm:ss
	
	■ **Dwell time**
	Countdown of the waiting time in seconds from the following functions:
	  ■ **FUNCTION DWELL**
	  ■ Cycle **9 DWELL TIME**
	  ■ Parameter **Q210 DWELL TIME AT TOP**
	  ■ Parameter **Q211 DWELL TIME AT DEPTH**
	  ■ Parameter **Q255 DWELL TIME**
	
	Further information: "Display of the program run time", Page 111

Programs called	Path of the main program as well as called NC programs including the path
Radius compensation	Programmed tool radius compensation

The POS tab

On the **POS** tab the control shows information about positions and coordinates.

Area	Contents
Position display (e.g., **Actual reference position (RFACTL)**)	In this area the control shows the current position of all axes that are present. You can choose between the following views in the position display: ■ **Nominal pos. (NOML)** ■ **Actual pos. (ACT)** ■ **Nominal reference position (RFNOML)** ■ **Actual reference position (RFACTL)** ■ **Servo lag (LAG)** ■ **Actual distance to go (ACTDST)** ■ **Nominal distance to go (REFDST)** **Further information:** "Position displays", Page 113
Feed and Speed	■ Active **Feed** in mm/min Only in the **Status** workspace If a feed rate limit is active, the control displays the line in orange. If the feed rate is limited using the **F LIMIT** button, the control displays **LIMIT** in square brackets. **Further information:** "Feed rate limit F LIMIT", Page 234 If the feed rate is limited by means of functional safety (FS), then the control displays the active safety function in brackets. **Further information:** "Safety functions", Page 321 ■ Active **Feed-rate override** in % Only in the **Status** workspace ■ Active **Rapid-traverse override** in % ■ Active **Programmed feed rate** in mm/min Only in the **Status** workspace If **M136** is active: active feed rate in mm/rev **Further information:** Programming and Testing User's Manual ■ Active **Spindle speed** in rpm ■ Active **Spindle override** in % Only in the **Status** workspace ■ Active **Miscellaneous function** in reference to the spindle, such as **M3**
Basic transformations	The control displays the values of the current workpiece preset in this area. **Further information:** "Preset management", Page 179
Active traverse ranges	Active traverse range, such as Limit 1 for traverse range 1 Traverse ranges are machine-specific. If no traverse range is active, then **Traverse range not defined** is displayed in this area.

Area	Contents
Active kinemat.	Name of the active machine kinematics

The POS HR tab

On the **POS HR** tab the control shows information about handwheel superimpositioning.

Area	Contents
Coordinate system	■ **Machine (M-CS)**
Handwheel superimp.	■ **Max. val.** ■ **Actl.val.** Current superimpositioning

The QPARA tab

On the **QPARA** tab the control shows information about the defined variables.

Further information: Programming and Testing User's Manual

Using the **Parameter list** and **Variable list** windows, you can define which variables the control shows in the areas. Up to 22 variables can be displayed in each area.

Further information: "Defining the contents of the QPARA tab", Page 115

Area	Contents
Q parameter	Shows the values of the selected Q parameters
QL parameter	Shows the values of the selected QL parameters
QR parameter	Shows the values of the selected QR parameters
QS parameter	Shows the contents of the selected QS parameters
Variables	Shows the contents of the selected named parameters

The Tables tab

On the **Tables** tab, the control shows information about the active tables for program run or the simulation.

Area	Contents
Active tables	In this area the control shows the path for the following active tables: ■ Tool table ■ Preset table ■ Datum table ■ Pocket table ■ Touch-probe table (#17 / #1-05-1)

The TRANS tab

On the **TRANS** tab the control shows information about active transformations in the NC program.

Area	Contents
Active datum	▪ Path of the selected datum table ▪ Row number of the selected datum table ▪ **DOC** Contents of the **DOC** column of the datum table
Active datum shift	Datum shift that was defined with the **TRANS DATUM** function **Further information:** Programming and Testing User's Manual
Mirrored axes	Axes mirrored with either the **TRANS MIRROR** function or Cycle **8 MIRRORING** **Further information:** Programming and Testing User's Manual
Center of scaling	Center of scaling that was defined with Cycle **26 AXIS-SPECIFIC SCALING** **Further information:** Programming and Testing User's Manual
Active scaling factors	Scaling factors that were defined for the individual linear axes with the **TRANS SCALE** function, Cycle **11 SCALING FACTOR** or Cycle **26 AXIS-SPECIFIC SCALING** **Further information:** Programming and Testing User's Manual **Further information:** Programming and Testing User's Manual

The TT tab

On the **TT** tab the control shows information about measurements performed with a TT tool touch probe.

Further information: "Hardware enhancements", Page 56

Area	Contents
TT: tool measurement	■ **T** Tool number ■ **Name** Tool name ■ **Measuring method** Selected measurement method for tool measurement (e.g., **Length**) ■ **Min (mm)** When measuring milling cutters, in this area the control shows the smallest measured value of a cutting edge. ■ **Max (mm)** When measuring milling cutters, in this area the control shows the greatest measured value of a cutting edge. ■ **DYN Rotation (mm)** When measuring milling cutters with a rotating spindle, the control shows values in this area.
TT: measurement of individual teeth	**Number** List of the measurements performed and the measured values of the individual cutting edges

The Tool tab

On the **Tool** tab, the control shows information about the active tool, depending on the tool type.

Further information: "Tool types", Page 161

Area	Contents
Tool information	■ **T** Tool number ■ **Name** Tool name ■ **Tool Axis** Tool axis programmed in the tool call (e.g., **Z**) ■ **Type** Tool type of the active tool (e.g., **DRILL**) ■ **DOC** Note on the tool
Tool geometry	■ **L** Tool length ■ **R** Tool radius ■ **R2** Corner radius of the tool
Tool allowances	■ **DL** Delta value for the tool length ■ **DR** Delta value for the tool radius ■ **DR2** Delta value for the corner radius of the tool With **Program**, the control displays the values from a tool call with **TOOL CALL** or from a tool compensation with a compensation table ***.tcs**. **Further information:** Programming and Testing User's Manual With **Table**, the control displays the values from the tool management. **Further information:** "Tool management ", Page 163
Tool ages	■ **Cur. time (min)** Current amount of time during which the tool has been engaged ■ **Time 1 (min)** Service life of the tool ■ **Time 2 (min)** Maximum service life at tool call
Replacement tool	■ **RT** Tool number of the replacement tool ■ **Name** Tool name of the replacement tool

4.5   The Simulation status workspace

Application

You can call additional status displays in the **Editor** operating mode in the **Simulation status** workspace. In the **Simulation status** workspace, the control shows data based on the simulation of the NC program.

Description of function

The following tabs are available in the **Simulation status** workspace:
■ **Favorites**

Further information: "The Favorites tab", Page 102
■ **CYC**

Further information: "CYC tab", Page 102
■ **FN 16**

Further information: "The FN 16 tab", Page 102
■ **LBL**

Further information: "The LBL tab", Page 103
■ **M**

Further information: "The M tab", Page 103
■ **PGM**

Further information: "The PGM tab", Page 104
■ **POS**

Further information: "The POS tab", Page 105
■ **QPARA**

Further information: "The QPARA tab", Page 106
■ **Tables**

Further information: "The Tables tab", Page 106
■ **TRANS**

Further information: "The TRANS tab", Page 107
■ **TT**

Further information: "The TT tab", Page 108
■ **Tool**

Further information: "The Tool tab", Page 109

Notes

■ The control displays the current counter reading and the defined target quantity only on the **PGM** tab of the **Status** workspace.

Further information: "The PGM tab", Page 104

Further information: Programming and Testing User's Manual
■ The following contents are only displayed on the **POS** tab of the **Status** workspace:
 ■ Active **Feed** in mm/min
 ■ Active **Feed-rate override** in %
 ■ Active **Spindle override** in %

Further information: "The POS tab", Page 105

4.6 Display of the program run time

Application

The control calculates the duration of all traverse movements and displays them together as the **Program run time**. The control takes traversing movements and dwell times into account.

In addition, the control calculates the remaining run time of the NC program.

Description of function

The control displays the program run time in the following areas:

- **PGM** tab of the **Status** workspace
  Further information: "The PGM tab", Page 104
- Status overview on the TNC bar
  Further information: "Status overview of the TNC bar", Page 99
- **PGM** tab of the **Simulation status** workspace
- The **Simulation** workspace in the **Editor** operating mode
  Further information: Programming and Testing User's Manual

Use the **Settings** button in the **Program run time** area to influence the calculated program run time.

The control opens a selection menu with the following functions:

Function	Meaning
Save	Save the current value under **Runtime**
Addition	Add the saved time to the value under **Runtime**
Resetting	Reset the saved time and the contents of the **Program run time** area to zero

The control counts the time during which the **Control-in-operation** symbol is green. The control adds the time from the **Program Run** operating mode and the **MDI** application.

The following functions reset the program run time:

- Selecting a new NC program for program run
- The **Reset program** button
- The **Resetting** function in the **Program run time** area

Remaining run time of the NC program

If a tool usage file is available, the control calculates for the **Program Run** operating mode the duration of executing the active NC program. During program run, the control updates the remaining run time.

Further information: "Tool usage test", Page 169

The control shows the remaining run time in the status overview on the TNC bar.

The control does not take the feed-rate potentiometer setting into account, but calculates with a feed rate of 100%.

The following functions reset the remaining run time:

- Selecting a new NC program for program run
- **Internal stop** button
- Generate new tool usage file

Notes

- In the machine parameter **operatingTimeReset** (no. 200801) the machine manufacturer defines whether the control resets the program run time when the program is started.
- The control cannot simulate the run time of machine-specific functions such as tool changing. That is why this function is only partially suitable for calculating the production time in the **Simulation** workspace.
- In the **Program Run** operating mode, the control displays the exact time of the NC program while taking all machine-specific actions into account.

Definition

Control-in-operation:
The control uses the **Control-in-operation** symbol to show the machining status of the NC program or NC block:

- White: no movement command
- Green: active machining, axes are moving
- Orange: NC program interrupted
- Red: NC program stopped

Further information: "Interrupting, stopping or canceling program run", Page 235

When the control bar is expanded, the control shows additional information about the current status, such as **Active, feed rate at zero**.

4.7 Position displays

Application

The control offers various modes in the position display, for example values from different reference systems. You can choose one of the modes available based on the application.

Description of function

The control has position displays in the following areas:

- The **Positions** workspace
- Status overview on the control bar
- The **POS** tab of the **Status** workspace
- The **POS** tab of the **Simulation status** workspace

On the **POS** tab of the **Simulation status** workspace the control always shows the **Nominal pos. (NOML)** mode. In the **Status** and **Positions** workspaces you can choose the mode of the position display.

The control offers the following modes for the position display:

Mode	Meaning
Nominal pos. (NOML)	This mode shows the value of the currently calculated target position in the input coordinate system **I-CS**.
	When the machine moves the axes, the control compares the coordinates of the measured actual position with the calculated nominal position in predefined time intervals. The nominal position is the position at which the axes should be located at the time of comparison, based on the calculation.
	ℹ The **Nominal pos. (NOML)** and **Actual pos. (ACT)** modes differ solely with regard to the servo lag.
Actual pos. (ACT)	This mode shows the currently measured tool position in the input coordinate system **I-CS**.
	The actual position is the measured position of the axes, as determined by encoders at the time of comparison.
Nominal reference position (RFNOML)	This mode shows the calculated target position in the machine coordinate system **M-CS**.
	ℹ The **Nominal reference position (RFNOML)** and **Actual reference position (RFACTL)** modes differ solely with regard to the servo lag.
Actual reference position (RFACTL)	This mode shows the currently measured tool position in the machine coordinate system **M-CS**.
Servo lag (LAG)	This mode shows the difference between the calculated nominal position and the measured actual position. The control determines the difference in predefined time intervals.

Mode	Meaning
Actual distance to go (ACTDST)	This mode displays for every axis the difference between the actual position and target position in the **I-CS** input coordinate system.
	The control takes all the coordinate transformations into account and displays the remaining distance to the programmed target position.
Nominal distance to go (REFDST)	This mode displays for every axis the difference between the actual position and the target position in the machine coordinate system **M-CS**.
	The control displays the distance that every physical axis actually moves and does this independently of the active coordinate transformations.

> Refer to your machine manual.
>
> In the machine parameter **progToolCallDL** (no. 124501), the machine manufacturer defines whether the position display takes the delta value **DL** from the tool call into account. The modes **NOML.** and **ACTL.** as well as **RFNOML** and **RFACTL** then differ from each other by the value **DL**.

4.7.1   Switching the position display mode

To switch the position display mode in the **Status** workspace:

▶ Select the **POS** tab

> ▶ Select **Settings** in the position display area
> ▶ Select the desired mode for the position display (e.g., **Actual pos. (ACT)**)
> > The control displays the positions in the selected mode.

Notes

- The machine parameter **CfgPosDisplayPace** (no. 101000) defines the display accuracy by the number of decimal places.
- If the axes are moved, then, in addition to the current positions, the control also displays the remaining distances-to-go with a symbol and the respective value.
  Exception: The **Actual distance to go (ACTDST)** and **Nominal distance to go (REFDST)** modes
  Further information: "Axis display and position display", Page 96

4.8 Defining the contents of the QPARA tab

On the **QPARA** tab of the **Status** and **Simulation status** workspaces, you can define which variables the control will show.

Further information: "The QPARA tab", Page 106

To define the contents of the **QPARA** tab:

▶ Select the **QPARA** tab

⚙   ▶ Select the **Settings** in the desired area, such as QL parameters

       > The control opens the **Parameter list** window.

       ▶ Enter the number or name (e.g., **1,3,200-208**)

    OK   ▶ Press **OK**

       > The control displays the values of the defined variables.

ⓘ
- Use commas to separate single variables and connect sequential variables with a hyphen.
- The control always shows eight decimal places on the **QPARA** tab. For example, the control shows the result of **Q1 = COS 89.999** as 0.00001745. Very large and very small values are shown in exponential notation. The control shows the result of **Q1 = COS 89.999 * 0.001** as +1.74532925e-08, with e-08 corresponding to the factor of 10^{-8}.
- If you check the content of a string parameter on the **QPARA** tab of the **Status** workspace, you possibly do not see the complete content.

5

Powering on and off

5.1   Switch-on

Application

After using the main switch to power on the machine, the control's boot process begins. The following steps may differ depending on the machine; for example, whether absolute or incremental position encoders are used.

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

Related topics

■ Absolute and incremental position encoders

Further information: "Position encoders and reference marks", Page 147

Description of function

> ⚠ **DANGER**
>
> **Caution: hazard to the user!**
>
> Machines and machine components always pose 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

Power-on of the control begins with the power supply.

After booting, the control checks the machine status, e.g.:

■ Positions identical to before switching off the machine
■ Safety features are ready, such as the emergency stop
■ Functional safety

If the control registers an error during or after booting, it issues an error message.

The following step differs depending on position encoders on the machine:

■ Absolute position encoders

 If the machine has absolute position encoders, the control opens the **Start/Login** application after power-on.

■ Incremental position encoders

 If the machine has incremental position encoders, you must traverse the reference points in the **Move to ref. point** application. Once all axes have been referenced, the control is in the **Manual operation** application.

 Further information: "The Referencing workspace", Page 120

 Further information: "The Manual operation application", Page 124

5.1.1 Powering the machine and the control on

To switch the machine on:

▶ Switch on the power supply of the control and of the machine
> The control is in start-up mode and shows the progress in the **Start** workspace.
> The control displays the **Power interrupted** dialog in the **Start** workspace.

| OK | ▶ Select **OK** |
| | > The control compiles the PLC program. |

I	▶ Switch the machine control voltage on
	> The control checks the functioning of the emergency stop circuit.
	> If the machine is equipped with absolute linear and angle encoders, the control is now ready for operation.
	> If the machine is equipped with incremental linear and angle encoders, the control opens the **Move to ref. point** application.

Further information: "The Referencing workspace", Page 120

⟲	▶ Press the **NC Start** key
	> The control moves to all necessary reference points.
	> The control is ready for operation and the **Manual operation** application is open.

Further information: "The Manual operation application", Page 124

ℹ If startup is delayed due to functional safety, the control displays the text **Functional safety requires input**. When you select the **FS** button, the control switches to the **Functional safety** application.

Further information: "The Functional safety application", Page 322

Notes

NOTICE
Danger of collision!
Failure to notice deviations between the actual axis positions and those expected by the control (saved at shutdown) can lead to undesirable and unexpected axis movements. There is risk of collision during the reference run of further axes 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 manufacturer

■ Once startup has been completed, the control will close the **Start** workspace.

5.2 The Referencing workspace

Application

On machines with incremental linear and angle encoders, the control shows in the **Referencing** workspace which axes need to be referenced.

Description of function

The **Referencing** workspace is always open in the **Move to ref. point** application. If reference points are to be traversed when powering-on the machine, then the control opens this application automatically.
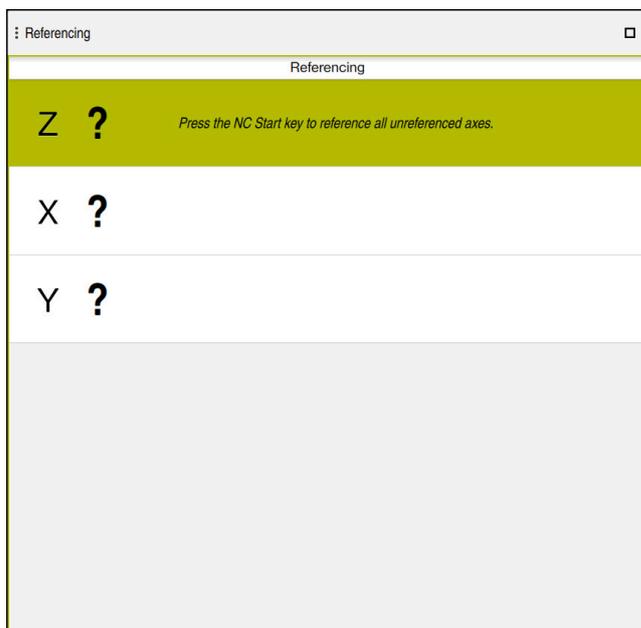
The **Referencing** workspace with axes to be referenced

The control displays a question mark behind all axes that need to be referenced.

Once all axes have been referenced, the control closes the **Move to ref. point** application and switches to the **Manual operation** application.

5.2.1 Axis reference run

To reference the axes in the prescribed sequence:

▶ Press the **NC start** key
> The control moves to the reference points.
> The control switches to the **Manual operation** application.

To reference the axes in any sequence:

▶ Press and hold the axis direction button for each axis until the reference point has been traversed
> The control switches to the **Manual operation** application.

Notes

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

- You cannot switch to the **Program Run** operating mode as long as reference points still need to be traversed.
- If you intend only to edit or simulate NC programs, you can switch to the **Editor** operating mode without referencing the axes. You can still traverse the reference points at a later time.

5.3 Powering off

Application

To avoid losing data, shut down the control before powering-off the machine.

Description of function

You can shut down the control in the **Start/Login** application of the **Home** operating mode.

If you select the **Shut down** button, the control opens the **Shut down** window. You choose whether to shut down the control or restart it.

If NC programs or contours contain any unsaved changes, the control displays the unsaved changes in the **Close file** window. You can save the changes, discard them, or cancel the shutdown.

5.3.1 Shutting down the control and powering-off the machine

To power-off the machine:

⌂
> ▶ Select the **Home** operating mode

Shut down
> ▶ Select **Shut down**
> > The control opens the **Shut down** window.
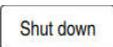
Shut down
> ▶ Select **Shut down**
> > If NC programs or contours contain any unsaved changes, the control displays the **Close file** window.
> ▶ If necessary, save unsaved NC programs with **Save** or **Save as**
> > The control shuts down.
> > After completion of the shutdown process, the control displays the text **Now you can switch off.**
> ▶ Switch off the main power switch of the machine

Notes

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 regardless of the control's status!
▶ Always shut down the control
▶ Only operate the main switch after being prompted on the screen

■ Different machines have different power-off procedures.
Refer to your machine manual.

6

Manual operation

6.1 The Manual operation application

Application

In the **Manual operation** application, you can move the axes manually, for example, and set up the machine.

Related topics

- Moving the machine axes
  Further information: "Moving the machine axes", Page 126
- Incremental jog positioning of machine axes
  Further information: "Incremental jog positioning of axes", Page 129

Description of function

The **Manual operation** application offers the following workspaces:
- **Fixed cycles**
- **Document**
- **Graphic Positioning**
- **Positions**
- **Simulation**
- **Status**

The function bar in the **Manual operation** application contains the following buttons:

Button	Meaning
Fixed cycles	The control opens the **Fixed cycles** workspace.
	Further information: "The Fixed cycles workspace", Page 132
M	Define a miscellaneous function **M** or use the selection menu to choose one and activate it with the **NC start** key.
	Further information: Programming and Testing User's Manual
	The machine manufacturer uses the optional machine parameter **forbidManual** (no. 103917) to define which miscellaneous functions are allowed in the **Manual operation** application and are available in the selection menu.
S	Define the spindle speed **S**, activate it with the **NC start** key, and also switch on the spindle.
	Further information: Programming and Testing User's Manual
F	Define the feed rate **F** and activate it with the **OK** button.
	Further information: Programming and Testing User's Manual
T	Define a tool **T** or use the selection window to choose one and insert it with the **NC start** key.
	Further information: Programming and Testing User's Manual
Active preset	The control opens the preset table in the **Active preset** window.
	Further information: "Preset table *.pr", Page 295
Q info	The control opens the **Q parameter list** window, where you can see and edit the current values and descriptions of the variables.
	Further information: Programming and Testing User's Manual
Manual cycles	The machine manufacturer can define manual cycles that you can use by means of this button.

Button	Meaning
Jog increment	Define the jog increment **Further information:** "Incremental jog positioning of axes", Page 129
Set the preset	Enter and set a preset **Further information:** "Preset management", Page 179
Position the axis manually	The control opens the **Position the axis manually** window in which you define a target position for the axis. You can either move the axis automatically or position it manually. **Further information:** "The Position the axis manually window", Page 127
Tools	The control opens the **Tool management** application in the **Tables** operating mode. **Further information:** "Tool management ", Page 163
Handwheel	The control displays this toggle switch if a handwheel without display is configured on the control. If the handwheel is active, the operating mode's icon in the sidebar changes. **Further information:** "Electronic handwheel", Page 301
Internal stop	For example, if an NC program is interrupted due to an error or a stop, the control activates this button. Use this button to abort program run. **Further information:** "Interrupting, stopping or canceling program run", Page 235

6.2 Moving the machine axes

Application

You can use the control to move the machine axes manually, such as pre-positioning for a manual touch probe function.

Further information: "Touch probe functions in the Manual operating mode (#17 / #1-05-1)", Page 209

Related topics

- Programming traverse movements
  Further information: Programming and Testing User's Manual
- Executing traverse movements in the **MDI** application
  Further information: "The MDI application ", Page 205

Description of function

The control offers the following methods for moving axes manually:

- Axis-direction keys
- Incremental jog positioning with the **Jog increment** button
- Traversing with electronic handwheels
  Further information: "Electronic handwheel", Page 301
- Move to the target position with the **Position the axis manually** window

The control displays the current contouring feed rate in the status display while the machine axes are in motion.

Further information: "Status displays", Page 93

You can change the contouring feed rate with the **F** button in the **Manual operation** application and with the feed-rate potentiometer.

A traverse job is active on the control as soon as an axis moves. The control shows the status of the traverse job with the **Control-in-operation** icon in the status overview.

Further information: "Status overview of the TNC bar", Page 99

The Position the axis manually window

You can define a target position in the **Position the axis manually** window. You can either manually move the axes to the target position or have them automatically moved in the case of controlled axes.
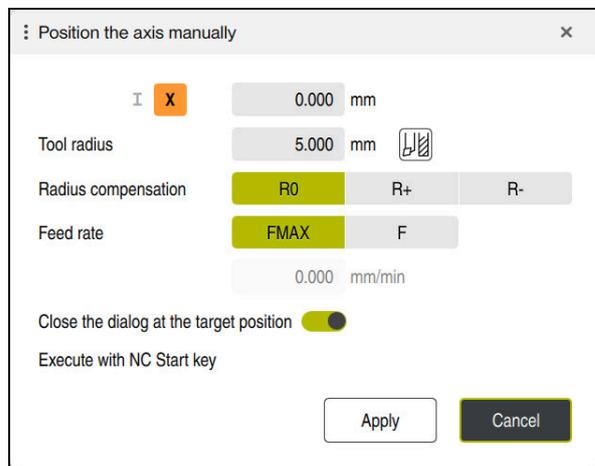
The **Position the axis manually** window

You can open the **Position the axis manually** window in the following ways:
- Select the **Position the axis manually** button
- Press the axis key
- Tap or click in the area of the desired axis in the **Positions** workspace

    If you tap or click on the orange axis designation, the control opens the **Presetting** window instead.

In the **Position the axis manually** window, you can define the target position with the following inputs:

Input	Meaning
I X	Define the target position If you additionally tap or click on **I**, then you can enter an incremental value.
Tool radius	Radius of the tool for radius compensation Selection can be performed via a selection window.

Radius compensation	■ **R0**
	  The control performs positioning in accordance with the tool center point.
	■ **R+**
	  The control lengthens the tool movement by the amount of the tool radius.
	■ **R-**
	  The control shortens the tool movement by the amount of the tool radius.

Feed rate	■ **FMAX**
	  The control positions the tool in rapid traverse.
	■ **F**
	  The control positions the tool at a defined feed rate (e.g., **3000**).

Input	Meaning
Close the dialog after positioning	If you activate the toggle switch, then the control closes the **Position the axis manually** window at the target position.

You can reach the target point in the following ways:

- By pressing the **NC Start** key

  In the case of controlled axes, the control automatically moves to the defined target position.

  Further information: "Automatically moving the axis to the target position", Page 130

- By pressing the **Apply** button

  The control displays a positioning scale below the position display in the **Positions** workspace.

  You can move manual or controlled axes to the target position with the handwheel, for example. If the orange square is in the middle of the position scale, then the axis is at the target position.

  Further information: "Automatically moving the axis to the target position", Page 131

6.2.1 Using axis keys to move the axes

To move an axis manually with the axis keys:

🖐	▶ Select an operating mode (e.g., **Manual**)
	▶ Select an application (e.g., **Manual operation**)
X+	▶ Press the axis key of the desired axis
	> The control moves the axis as long as you press the key.

> ⓘ If you hold the axis key pressed down and simultaneously press the **NC Start** key, the control moves the axis at a continuous feed rate. You have to end traverse movement with the **NC Stop** key.

6.2.2 Incremental jog positioning of axes

Incremental jog positioning allows you to move a machine axis by a preset distance. The input range for the infeed is from 0.001 mm to 10 mm.
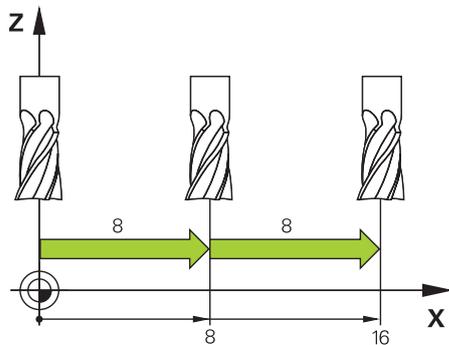
To position an axis incrementally:

🖑	▶ Select the **Manual** operating mode
🖑	▶ Select the **Manual operation** application
Jog increment	▶ Select **Jog increment**
	> The control opens the **Positions** workspace, if necessary, and shows the **Jog increment** area.
	▶ Enter the jog increment for linear axes
X+	▶ Press the axis key of the desired axis
	> The control positions the axis in the selected direction by the defined jog increment.
Off	▶ Select **Off**
	> The control ends incremental jog positioning and closes the **Jog increment** area in the **Positions** workspace.

> 🛈 You can also end incremental positioning with the **Jog inc. on** button.
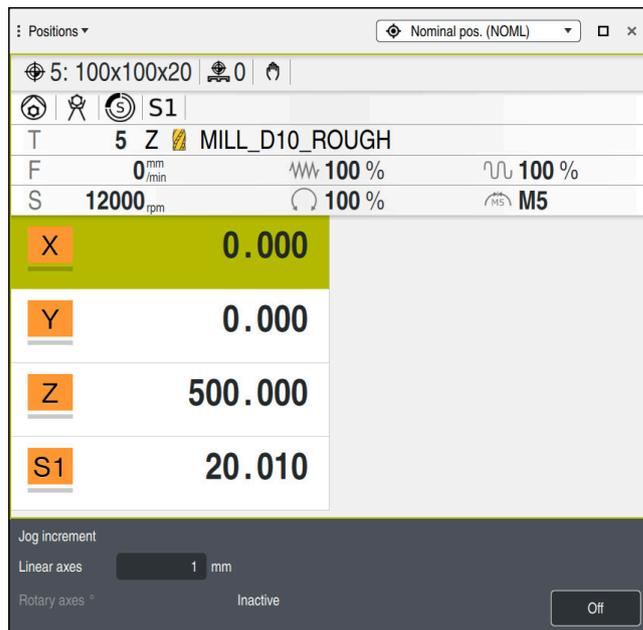
The **Positions** workspace with active **Jog increment** area

6.2.3   Automatically moving the axis to the target position

To move a controlled axis to the defined target position:

▶ Select the **Manual** operating mode

▶ Select the **Manual operation** application

▶ Select **Position the axis manually**

> The control opens the **Position the axis manually window.**

▶ Select the other axis in the **Positions** workspace as needed

▶ Enter the target position (e.g., **40**)

▶ Tap or click on **I** as needed

▶ Enter the **Tool radius** as needed (e.g., **4**)

▶ Define the **Radius compensation** as needed (e.g., **R+**)

▶ Enter the **Feed rate** (e.g., **3000**)

▶ Press the **NC Start** key

> The control positions the axis.

6.2.4 Automatically moving the axis to the target position

To move a manual or controlled axis to the defined target position:

▶ Select the **Manual** operating mode

▶ Select the **Manual operation** application

Position the axis manually

▶ Select **Position the axis manually**

> The control opens the **Position the axis manually window.**

▶ Select the other axis as in the **Positions** workspace as needed

▶ Enter the target position (e.g., **40**)

▶ Tap or click on **I** as needed

▶ Enter the **Tool radius** as needed (e.g., **4**)

▶ Define the **Radius compensation** as needed (e.g., **R+**)

Apply

▶ Select **Apply**

> The control displays a positioning scale below the position display in the **Positions** workspace.

▶ Move the axis to the middle of the position scale (e.g., with the handwheel)

OK

▶ Select **OK**

> The control closes the **Position the axis manually** window.

Notes

- When positioning an axis, the control checks whether the defined speed has been reached. The control does not check the speed in positioning blocks where **FMAX** is the feed rate.

- With the optional machine parameter **userControl** (no. 144101), the machine manufacturer defines whether you can select the operating element with which you move the axes to the target position.

  Further information: "Oper. station for program interruption", Page 334

Definition

Manual axis

Manual axes are non-driven axes that need to be positioned by the machine operator.

6.3 The Fixed cycles workspace

Application

In the **Fixed cycles** workspace, you can define a cycle the control will execute at the current position.

> ⚙ Refer to your machine manual.
> - The machine manufacturer defines the HEIDENHAIN cycles available to you.
> - The machine manufacturer can also provide customer cycles.

Related topics

- Cycles for drilling and boring
- Cycles for milling
  Further information: Programming and Testing User's Manual
- Executing a cycle at multiple positions in the **Graphic Positioning** workspace
  Further information: "The Graphic Positioning workspace", Page 135

Description of function

The **Fixed cycles** workspace is available in the **Manual operation** application.
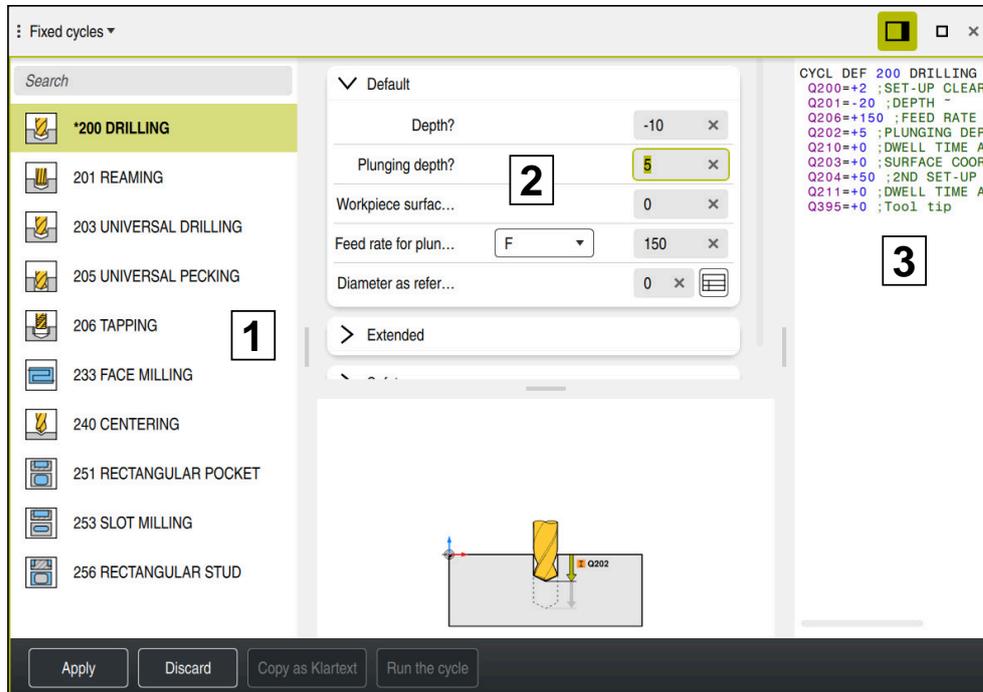
Columns of the **Fixed cycles** workspace

The **Fixed cycles** workspace contains the following columns:

1 Cycle selection column

   The control displays all available cycles.

   You can search for numbers or names in the **Search** input field.

2 Cycle parameters column

   Within the fillable form, the control groups the parameters based on their purpose (e.g., standard or safety).

   If an input is invalid, the control displays an information symbol. When you select the information symbol, the control displays information about the error.

3 The **Cycle preview** column

   The control displays a preview of the cycle with the currently defined parameters.

Icons and buttons

Icon or button	Meaning
▯	Show or hide the **Cycle preview** column
Apply	■ Confirm input ■ Update the cycle preview contents
Discard	Discard input
Copy as Klartext	Copy Cycle preview to the clipboard
Run the cycle	Open the **Run the cycle** window By pressing the **NC start** key, you can start the cycle, and the window will remain open during machining. Pressing **Cancel** closes the **Run the cycle** window.

6.3.1   Executing the cycle at the desired position

To execute the cycle at the desired position:

▶ Move the axes to the desired position
▶ Select the **Manual operation** application as needed
▶ Open the **Fixed cycles** workspace as needed
▶ Select the desired cycle
▶ Enter the input value
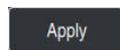
Apply
▶ Select **Apply**
> The control shows the current value in the cycle preview.
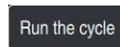
Run the cycle
▶ Select **Run the cycle**
> The control opens the **Run the cycle** window.

▶ Press the **NC Start** key
> The control executes the cycle at the current position.

6.3.2   Copying the cycle in Klartext

To copy the cycle and insert it into the **MDI** application:

▶ Select the desired cycle in the **Fixed cycles** workspace
▶ Enter the input value

Apply
▶ Select **Apply**
> The control shows the current value in the cycle preview.
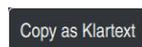
Copy as Klartext
▶ Select **Copy as Klartext**
> The control copies the cycle to the clipboard.

▶ Select the **MDI** application
▶ Select the desired NC block

▶ Select **Paste**
> The control inserts the cycle.

6.4 The Graphic Positioning workspace

6.4.1 Fundamentals

Application

In the **Graphic Positioning** workspace, you can define a cycle the control will execute at multiple positions. You can define the machining positions graphically.

> ⚙ Refer to your machine manual.
> - The machine manufacturer defines the HEIDENHAIN cycles available to you.
> - The machine manufacturer can also provide customer cycles.

Related topics

- Executing a cycle at the current position in the **Fixed cycles** workspace
  Further information: "The Fixed cycles workspace", Page 132
- Pattern definition with **PATTERN DEF**
- Cycles for drilling and boring
- Cycles for milling
  Further information: Programming and Testing User's Manual

Description of function

The **Graphic Positioning** workspace is available in the **Manual operation** application.

Screen layout
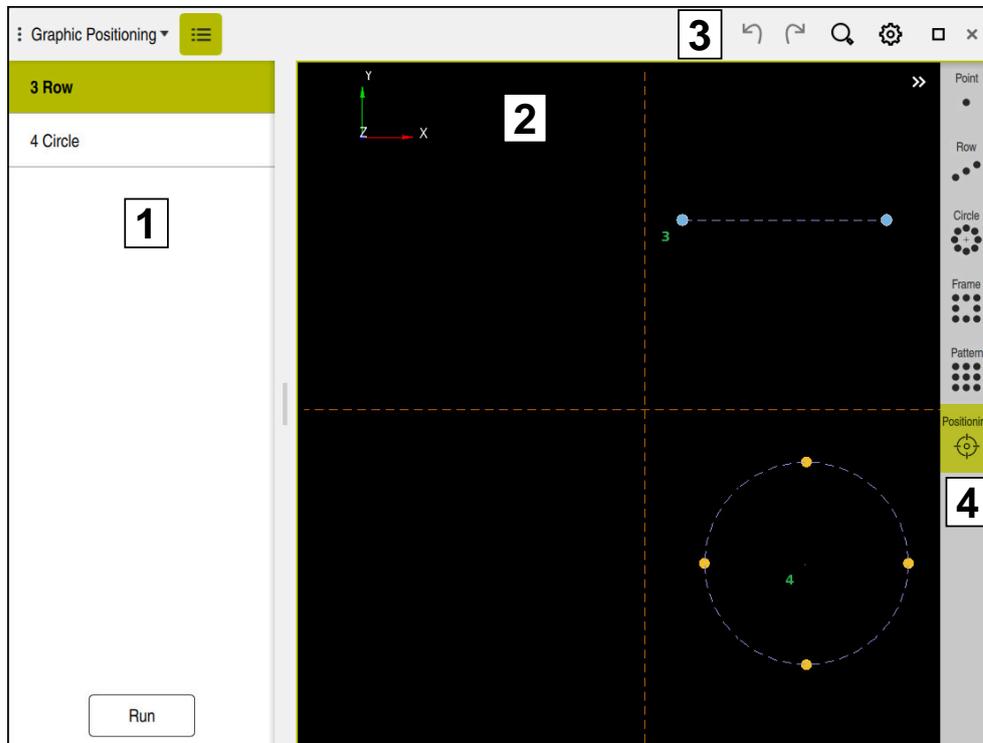
The screen layout of the **Graphic Positioning** workspace

The **Graphic Positioning** workspace contains the following areas:

1   The **List** column
2   The **Sketching** area
3   Title bar
4   Toolbar

Icons in the Graphic Positioning workspace

Title bar

The title bar of the **Graphic Positioning** workspace contains the following icons:

Icon or shortcut	Meaning
☰	Open or close the **List** column
▯ **[CTRL] + [N]**	**Discard** the hole pattern
🔍	Open or close the **Scaling options** selection menu
⌗	**Drawing area** Scale the view to the drawing area You can define the size of the drawing area in the **Settings for Graphic Positioning** window. **Further information:** "The Settings for Graphic Positioning window", Page 139
⊞	**Selected elements** Scale the view to the selected elements
⊞	**All elements** Scale the view to all elements
⚙	Open or close the **Settings for Graphic Positioning** window **Further information:** "The Settings for Graphic Positioning window", Page 139

Further information: "Icons on the control's user interface", Page 73

Toolbar

The control displays the following icons in the toolbar:

Icon	Meaning
•	Add and define a point
⁚•°	Define a row
⋮	Define a partial or complete circle
⦙⦙	Define a frame
⦙⦙⦙	Define a pattern
⨁	Select and execute a point pattern

Possible colors

The control shows the elements in the following colors:

Icon	Meaning
🟡 🔵 🟢	Point patterns ▪ Yellow: Standard ▪ Blue: Selected point ▪ Green: Active point in the **Position to point** window
▬ ▬	Auxiliary lines The control automatically adds auxiliary lines between multiple points (e.g., in the case of a **Frame**). Auxiliary lines are shown by the control as a blue dashed line.
▬ ▬	**Reference axis** The reference axes shown form a Cartesian coordinate system. The dimensions are relative to the intersection of the reference axes, which is equivalent to the workpiece preset. The control shows reference axes as brown dashed lines.

The List column

The **List** column with opened **Row** point pattern

In the **List** column, the control displays all inserted point patterns.

If you select the **Edit** icon, the control displays the parameters of the point pattern and the coordinates of the individual points. You can change the parameters.

If you select the **Positioning** icon and a point pattern, then the **Run** button is no longer dimmed.

Click the **Run** button to open the **Position to point** window.

Further information: "The Position to point window", Page 139

The Settings for Graphic Positioning window

The window contains the following settings:

Setting	Meaning
Sketching area width	Default width of the drawing area
Sketching area height	Default height of the drawing area
Decimal places	Number of decimal places

6.4.2 The Position to point window

Application

The **Position to point** window allows you to position the tool at the machining position and then define a cycle.

Related topics

- Executing the cycle at the current position in the **Fixed cycles** workspace
  Further information: "The Fixed cycles workspace", Page 132

Description of function

The Positioning area
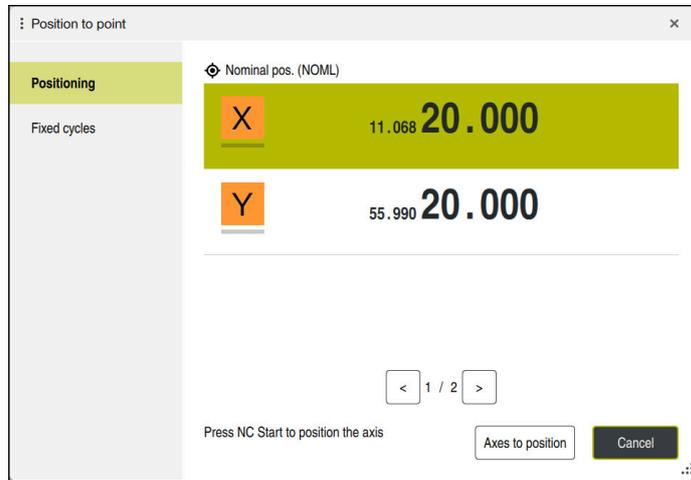
The **Positioning** area in the **Position to point** window

You can position the axes in the following ways:

- By pressing the **NC Start** key

  The control automatically moves the selected axis to the nominal position.

- Manually move the axis (e.g., with an axis key)

  You must continue pressing the axis key until the control reaches the position.

  In the case of the manual axes, the control displays a positioning scale for the most accurate positioning possible.

The **Positioning** area contains the following icons and buttons:

Icon or button	Meaning
‹ ›	**Navigate** If a point pattern contains multiple points (e.g., in the case of a **Row**), then you can select the desired point with the arrow keys. The control displays the selected point in the **Sketching** area.
Axes to position	Use the button to confirm that the axes are at the desired position. ⓘ The control does not automatically check the axes. The control switches to the **Fixed cycles** area.

The Fixed cycles area

This area is where you can select and define a machining cycle.

The **NC Start** key tells the control to execute the machining cycle at the current position.

> ℹ️ The control provides the same cycles for selection as in the **Fixed cycles** workspace.

The **Fixed cycles** area contains the following icons and buttons:

Icon or button	Meaning
⬜	Show or hide the **Help graphic** column
Fixed cycles	The control displays all available cycles in the selection menu.
Apply	Confirm input
Discard	Discard input
Next point	The control switches to the **Positioning** area in order to move the axes to the next point.

6.4.3 First steps in using the Graphic Positioning workspace

In this example task, you define a **Row** and execute a drilling cycle at the define points.

The following steps are required:

1 Defining a **Row** point pattern

2 Defining and executing the machining cycle

Example task D1472622

A-A

ø 4

5

15

20

80

20

A   A

95

80

744 650 A4

5

W

5

95

□ 100

3:10

				ID number		
Text:				Change No.	C000941-05	
				Phase:	Nicht-Serie	
⬡⊕	Original drawing		Platte	Werkstoff:		
	Scale	Format		Material:		
RoHS	1:1	A4	Plate			
Maße in mm / Dimensions in mm			Einzelteilzeichnung / Component Drawing	●blanke Flächen/Blank surfaces		

Werkstückkanten nach ISO 13715 Workpiece edges ISO 13715	Allgemeintoleranzen ISO 2768:1989-mH ≦6mm: ±0,2 General tolerances ISO 2768:1989-mH ≦6mm: ±0,2	Tolerierung nach ISO 8015 Tolerances as per ISO 8015	Oberflächen nach ISO 1302:2002 Surfaces as per ISO 1302:2002
-0.3 +0.3		Oberflächenbehandlung: Surface treatment:	

The reproduction, distribution and utilization of this document as well as the communication of its contents to others without express authorization is prohibited. Offenders will be held liable for the payment of damages. All rights reserved in the event of the grant of a patent, utility model or design. (ISO 16016)

HEIDENHAIN DR. JOHANNES HEIDENHAIN GmbH 83301 Traunreut, Germany	Created	Responsible	Released		Version	Revision	Sheet	Page
	M-TA			**D1472622-00-A-01**				1 of 1
	19.08.2025			Document number				

Defining a Row point pattern

▶ Select the **Graphic Positioning** workspace

▶ If applicable, open the **List** column

▶ Select **Row**

▶ In the **Sketching** area, tap or click on the desired position

> In the **Sketching** area, the control inserts a point and opens the parameters for the **Row**.

▶ Define the **Starting point X** (e.g., **20**)

▶ Select **ENT**

> The control moves the point within the **Sketching** area.

▶ Define the **Starting point Y** (e.g., **80**)

▶ Select **ENT**

> The control moves the point within the **Sketching** area.

▶ Define the **Number of operations** (e.g., **2**)

▶ Select **ENT**

> In the **Sketching** area, the control inserts the second point and the auxiliary line.

> In the **Points list** area, the control displays the coordinates of the individual points.

▶ Define the **Spacing of machining positions** (e.g., **60**)

▶ Select **ENT**

> The control moves the second point within the **Sketching** area.

▶ Select **Back**

> The control saves the point pattern.

Defining and executing the machining cycle

To define the drilling cycle and drill the holes:

▶ Select **Positioning**

▶ Tap or click on the defined **Row**

Run ▶ Select **Run**

> The control opens the **Position to point** window.

▶ Press the **NC Start** key to position the X axis

> When the X axis reaches the position, the control automatically selects the Y axis.

▶ Press the **NC Start** key to position the Y axis

Axes to position ▶ Select **Axes to position**

> The control switches to the **Fixed cycles** area.

▶ Select Cycle **200 DRILLING**

▶ Define the cycle parameters

▶ Press the **NC Start** key to execute the cycle

> The control executes the cycle at the current position.
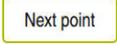
Next point ▶ Select **Next point**

> The control switches to the **Positioning** area.

▶ Press the **NC Start** key to position the X axis at the new point

> The control automatically selects the Y axis.

> The Y axis is already at the required position.
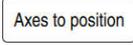
Axes to position ▶ Select **Axes to position**

> The control switches to the **Fixed cycles** area.

▶ Press the **NC Start** key to execute the cycle.

> The control executes the cycle at the second position.

Cancel ▶ Select **Cancel**

> The control closes the **Position to point** window.

7

NC fundamentals

7.1 NC fundamentals

7.1.1 Programmable axes
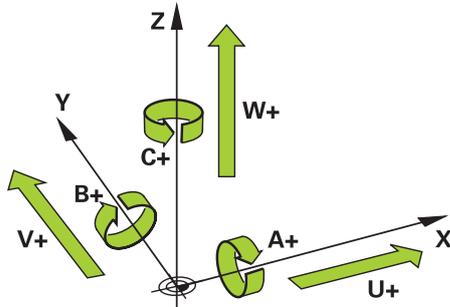
The programmable axes of the control are in accordance with the axis definitions specified in DIN 66217.

The programmable axes are designated as follows:

Main 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 manufacturer can define further axes, such as PLC axes.

> ℹ The TNC7 go can move up to four axes simultaneously. If an NC block commands movement of more than four axes, the control displays an error message.
>
> If the axis position does not change, you can nevertheless program more than four axes.

7.1.2 Designation of the axes of milling machines

The axes **X**, **Y** and **Z** on your machine are also designated as the main axis (1st axis), secondary axis (2nd axis) and tool axis. The main axis and secondary axis define the working plane.

The axes are associated as follows:

Main axis	Secondary axis	Tool axis	Working plane
X	Y	Z	**XY**, also **UV**, **XV**, **UY**
Y	Z	X	**YZ**, also **WU**, **ZU**, **WX**
Z	X	Y	**ZX**, also **VW**, **YW**, **VZ**

> ⚙ The control's full range of functions is available only if the **Z** tool axis is used (e.g., **PATTERN DEF**).
>
> Restricted use of the tool axes **X** and **Y** is possible when prepared and configured by the machine manufacturer.

7.1.3 Position encoders and reference marks

Fundamentals
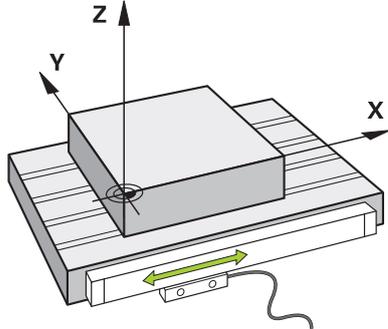
The position of the machine axes is ascertained with position encoders. As a rule, linear axes are equipped with linear encoders.

The position encoders detect the positions of the tool or machine table by generating an electrical signal during movement of an axis. The control ascertains the position of the axis in the current reference system from this electrical signal.

Further information: "Reference systems", Page 172

Position encoders can measure these positions through different methods:

■ Absolutely
■ Incrementally

The control cannot determine the position of the axes while the power is interrupted. Absolute and incremental position encoders behave differently once power is restored.

Absolute position encoders

On absolute position encoders, every position on the encoder is uniquely identified. The control can thus immediately determine the association between the axis position and the coordinate system after a power interruption.

Incremental position encoders

Incremental position encoders need to find the distance between the current position and a reference mark in order to determine the actual position. Reference marks indicate a machine-based reference point. A reference mark must be traversed in order to determine the current position after a power interruption.

If the position encoders feature distance-coded reference marks, then you need to move the linear encoders of the axes by no more than 20 mm.

Further information: "Axis reference run", Page 120
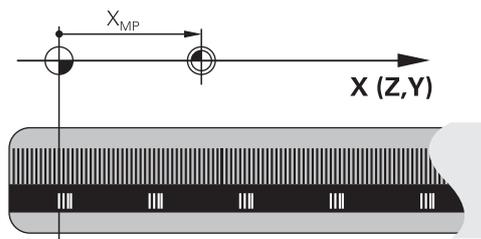
7.1.4 Presets in the machine

The following table contains an overview of the presets in the machine or on the workpiece.

Related topics

■ Presets on the tool

Further information: "Presets on the tool", Page 151

Icon	Preset
	Machine datum
	The machine datum is a fixed point defined in the machine configuration by the machine manufacturer.
	The machine datum is the origin of the machine coordinate system **M-CS**.
	Further information: "Machine coordinate system M-CS", Page 174
	If you program **M91** in an NC block, the defined values are referenced to the machine datum.
	Further information: Programming and Testing User's Manual
M92-ZP	**M92 datum M92-ZP (zero point)**
	The **M92** datum is a fixed point defined relative to the machine datum by the machine manufacturer in the machine configuration.
	The **M92** datum is the origin of the **M92** coordinate system. If you program **M92** in an NC block, the defined values are referenced to the **M92** datum.
	Further information: Programming and Testing User's Manual
	Tool change position
	The tool change position is a fixed point defined relative to the machine datum by the machine manufacturer in the tool-change macro.
	Reference point
	The reference point is a fixed point for initializing position encoders.
	Further information: "Position encoders and reference marks", Page 147
	If the machine has incremental position encoders, the axes must traverse the reference point after booting.
	Further information: "Axis reference run", Page 120
	Workpiece preset
	With the workpiece preset you define the origin of the workpiece coordinate system **W-CS**.
	Further information: "Workpiece coordinate system W-CS", Page 176
	The workpiece preset is defined in the active row of the preset table. You determine the workpiece preset with a 3D touch probe, for example.
	Further information: "Preset management", Page 179
	If no transformations are defined, the entries in the NC program refer to the workpiece preset.
	Workpiece datum
	You define the workpiece datum with transformations in the NC program, for example with **TRANS DATUM** or a datum table. The entries in the NC program refer to the workpiece datum. If no transformations are defined in the NC program, the workpiece datum corresponds to the workpiece preset.

8

Tools

8.1   Fundamentals

To use all of the control's functions, you must define the tools for the control using real data (e.g., the radius). This increase process reliability.

To add a tool to the machine and then be able to use it, follow the sequence below:

■ Clamp the tool in an appropriate tool holder.

■ To measure the tool dimensions, starting from the tool carrier preset, measure the tool (e.g., using a tool presetter). The control needs these dimensions for calculating the paths.

Further information: "Tool carrier reference point", Page 151

■ Further parameters are needed to completely define the tool. One place to find these parameters is the manufacturer's tool catalog.

Further information: "Tool parameters", Page 154

■ Save all collected parameters of this tool in the tool management.

Further information: "Tool management ", Page 163

■ As needed, assign a tool carrier to the tool in order to achieve realistic simulation and collision protection.

Further information: "Tool carrier management", Page 165

■ After finishing tool definition, program a tool call within an NC program.

Further information: Programming and Testing User's Manual

■ If your machine is equipped with a chaotic tool changer system and a double gripper, the tool change time may be shortened by pre-selecting the tool.

Further information: Programming and Testing User's Manual

■ If needed, perform a tool usage test before starting the program. This process checks if the tools are available in the machine and have sufficient remaining tool life.

Further information: "Tool usage test", Page 169

■ After machining a workpiece and measuring it, you may correct the tools.

Further information: Programming and Testing User's Manual

8.2 Presets on the tool

The control distinguishes the following presets on the tool for different calculations or applications.

Related topics

- Presets in the machine or on the workpiece

  Further information: "Presets in the machine", Page 148

8.2.1 Tool carrier reference point
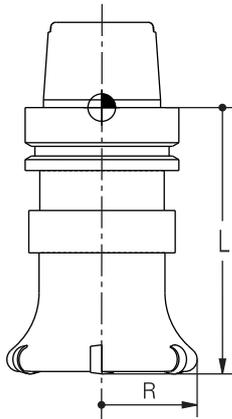
Milling cutter

The tool carrier reference point is a fixed point defined by the machine manufacturer. The tool carrier reference point is usually located on the spindle nose.

Starting from the tool carrier reference point, define the tool dimensions in the tool management (e.g., length **L** and radius **R**).

Further information: "Tool management ", Page 163

Further information: "Measuring the tool by scratching", Page 225

8.2.2 Tool tip TIP
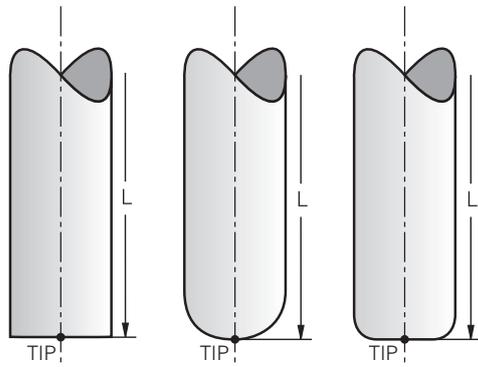
Milling tools

Further information: "Tool coordinate system T-CS", Page 178

You define the position of the tool tip with the basic and delta values of the tool relative to the tool-carrier reference point.

Further information: "Tool parameters", Page 154

In the case of milling cutters, the tool tip is located at the center of the tool diameter and at the longest point of the tool along the tool axis.

The tool tip is an auxiliary point for illustration purposes. The coordinates in the NC program reference the tool location point.

Further information: "Tool location point (TLP, tool location point)", Page 153

8.2.3 Tool center point (TCP, tool center point)
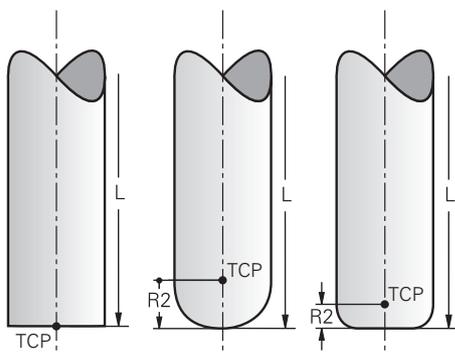
Milling tools

The tool center point TCP is the center of the tool diameter. If a tool radius 2 **R2** is defined, the tool center point is offset from the tool tip by this value.

Making entries in the tool management relative to the tool carrier reference point defines the tool center point.

Further information: "Tool management ", Page 163

The tool center point is an auxiliary point for illustration purposes. The coordinates in the NC program reference the tool location point.

Further information: "Tool location point (TLP, tool location point)", Page 153

8.2.4 Tool location point (TLP, tool location point)
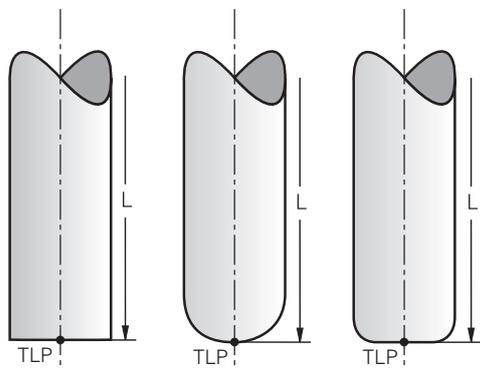
The control positions the tool on the tool location point TLP. By default, the tool location point is at the tool tip.

8.3 Tool parameters

Application

In the tool parameters you give the control all the information it needs to, for example, calculate the contours or run the simulation.

The parameters required depend, for example, on the tool type.

Related topics

- Editing parameters in tool management
  Further information: "Tool management ", Page 163
- Tool types
  Further information: "Tool types", Page 161
- Tool tables
  Further information: "Tool tables", Page 278
- Tool table **tool.t**
  Further information: "Parameters of the tool table tool.t", Page 279

Description of function

There are various possibilities for determining the parameters. For example:

- You can measure your tools in the machine (e. g., with a tool touch probe) or externally with a tool presetter.
  Further information: Programming and Testing User's Manual
- Take further tool information from the manufacturer's tool catalog (e.g., the material or the number of teeth).

The **Form** workspace in the **Tables** operating mode assists you when you enter the parameters. In the fillable form, the control filters the parameters in accordance with the selected tool types.

HEIDENHAIN recommends entering all known parameters in order to use the following functions to their full extent:

- Simulation
  Further information: Programming and Testing User's Manual
- Machining or touch probe cycles
  Further information: Programming and Testing User's Manual

8.3.1 Tool ID number

Application

Each tool has a unique number which equals the row number of the tool management. Each tool ID number is unique.

Further information: "Tool management ", Page 163

Description of function

The tool ID numbers can be defined in a range from 0 to 32,767.

The tool with the number 0 is defined as the zero tool, with both the length and radius being equal to 0. Upon a TOOL CALL 0, the control unloads the currently used tool and inserts no new tool.

Further information: Programming and Testing User's Manual

8.3.2 Tool name

Application

A tool name can be assigned in addition to the tool ID number. Contrary to the tool ID number, a tool name is not unique.

Description of function

The tool name allows identifying tools easier within the tool management. To this end, key features can be defined such as the diameter or the type of machining (e.g., **MILL_D10_ROUGH**).

As tool names are not unique, assign names that clearly identify the tools.

A tool name may contain up to 32 characters.

Permitted characters

You can use the following characters for the tool name:

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 # $ % & , - _ .

When entering lowercase letters, the control replaces them with uppercase letters upon saving.

Note

- Assign unique tool names!

    If you define identical tool names for multiple tools, the control will look for the tool in the following sequence:

    - Tool that is in the spindle
    - Tool that is in the magazine

    > ⚙ Refer to your machine manual.
    > If there are multiple magazines, the machine manufacturer can specify the search sequence of the tools in the magazines.

    - Tool that is defined in the tool table but is currently not in the magazine

    If the control, for example, finds multiple available tools in the tool magazine, it inserts the tool with the least remaining tool life.

8.3.3 Database ID

Application

In a tool database for all machines, you can identify tools with unique database IDs (e.g., within one machine shop). This allows you to coordinate the tools of multiple machines more easily.

The database ID is entered in the **DB_ID** column of the tool management.

Related topics

■ **DB_ID** column of tool management

   Further information: "Tool table tool.t", Page 278

Description of function

The database ID is stored in the **DB_ID** column of the tool management.

For indexed tools, you can define the database ID either only for the physically existing main tool or as an ID for the data record at each index.

For indexed tools, HEIDENHAIN recommends that you assign the database ID to the main tool.

Further information: "Indexed tool", Page 157

A database ID may contain a maximum of 40 characters and is unique in the tool management.

The control does not allow a tool call with the database ID.

8.3.4 Indexed tool

Application

Using an indexed tool, several different parameters can be stored for one physically available tool. This feature enables indication of a certain point on the tool by means of the NC program which does not necessarily have to correspond with the maximum tool length.

Requirement

- Main tool has been defined

Description of function

Tools with multiple lengths and radii cannot be defined in one row of the tool management table. Additional table rows are required, specifying the full definitions of the indexed tools. The lengths of the indexed tools, starting from the maximum tool length, approach the tool carrier preset as the index increases.

Further information: "Tool carrier reference point", Page 151

Further information: "Creating an indexed tool", Page 158

Examples of an application of indexed tools:

- Step drill

  The parameters of the main tool contain the drill tip, which corresponds to the maximum length. The tool steps are defined are indexed tools. This makes the lengths equal the actual tool dimensions.

- NC center drill

  The main tool is used for defining the theoretical tool tip as the maximum length. This can be used for centering, for example. The indexed tool defines a point along the tool tooth. This can be used for deburring, for example.

- Cut-off milling cutter or T-slot milling cutter

  The main tool is used for defining the lower point of the cutting edge, which equals the maximum length. The indexed tool defines the upper point of the cutting edge. When using the indexed tool for cutting-off, the specified workpiece height can be directly programmed.

Creating an indexed tool

To create an indexed tool:

▶ Select the **Tables** operating mode

▶ Select the **Tool management** application

▶ Enable **Edit**
> The control enables tool management for editing.
▶ Select **Insert tool**
> The control opens the **Insert tool** window.
▶ Select the desired tool type
▶ Define the tool number of the main tool (e.g., **T5**)
▶ Press **OK**
> The control adds table row **5**.
▶ Open the **Form** workspace
▶ Define all possible parameters in the form, including the maximum tool length
Further information: "Tool parameters", Page 154

▶ Select **Insert tool**
> The control opens the **Insert tool** pop-up window.
▶ Enable the **Index** check box
> The control adds the next free index number for the currently selected tool (e.g., **T5.1**).
▶ Press **OK**
> The control inserts table row **5.1** with the parameters of the main tool.
▶ Correct all deviating parameters in the form
Further information: "Tool parameters", Page 154

> ⓘ   The lengths of the indexed tools approach the tool carrier preset as the index rises, starting from the maximum tool length.
>
> **Further information:** "Tool carrier reference point", Page 151

Notes

- The control describes some parameters automatically, for example the current tool age **CUR_TIME**. The control describes these parameters separately for each table row.

  Further information: "Tool table tool.t", Page 278

- When you create an indexed tool, the control will copy the parameters from the previous table row. The previous table row can either be the main tool or an existing indexed tool.

- Index numbers do not need to be sequential. It is possible, for example, to create the tools **T5**, **T5.1** and **T5.3**.

- If you delete a main tool, the control will delete all associated indexed tools as well.

- If you copy or cut indexed tools only, you can use **Append** to add the indices to the currently selected tool.

  Further information: Programming and Testing User's Manual

- Up to nine indexed tools can be added to each main tool.

- If you define a replacement tool **RT**, this applies to the respective table row exclusively. When an indexed tool is worn and consequently blocked, this also does not apply to all other indices. This means, for example, that the main tool can still be used.

Example of T-slot milling cutter
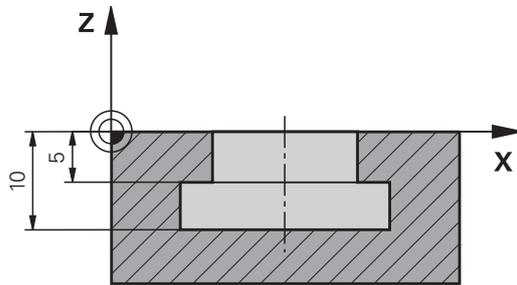
In this example, you program a T-slot with dimensions referring to the top and bottom edges as viewed from the coordinates surface. The height of the T-slot is larger than the length of the cutting edge of the tool used. This requires two steps.

Two tool definitions are required for producing the T-slot.

- The main tool dimension refers to the lower point of the cutting edge, which equals the maximum tool length. This can be used for machining the bottom edge of the T-slot.
- The dimension of the indexed tool refers to the upper point of the cutting edge. This can be used for machining the top edge of the T-slot.

> Please ensure that all required parameters are defined both for the main tool and for the indexed tool! In case of a rectangular tool, the radius remains identical in both table rows.

The T-slot is programmed in two machining steps:

- The 10 mm depth is programmed with the main tool.
- The 5 mm depth is programmed with the indexed tool.

11 TOOL CALL 7 Z S2000	; Call the main tool
12 X+0 R0 FMAX	; Pre-position the tool
13 Y+0 R0 FMAX	
14 Z+10 R0 FMAX	
15 Z-10 R0 F500	; Move to machining depth
16 CALL LBL "CONTOUR"	; Machine the bottom edge of the T-slot with the main tool
* - ...	
21 TOOL CALL 7 .1 Z S2000	; Call the indexed tool
22 X+0 R0 FMAX	; Pre-position the tool
23 Y+0 R0 FMAX	
24 Z+10 R0 FMAX	
25 Z-5 R0 F500	; Move to machining depth
26 CALL LBL "CONTOUR"	; Machine the top edge of the T-slot with the indexed tool

8.3.5 Tool types

Application

The control uses the tool types to filter the parameters that you can edit in tool management.

Related topics

- Editing parameters in tool management
  Further information: "Tool management ", Page 163
- Tool tables
  Further information: "Tool tables", Page 278

Description of function

A number is additionally assigned to each tool type.

The following tool types can be selected using the **TYP** parameter of the tool management:

Icon	Tool type	Number
	Milling cutter (**MILL**)	0
	Rough cutter (**MILL_R**)	9
	Finishing cutter (**MILL_F**)	10
	Face mill (**MILL_FACE**)	14
	Ball-nose cutter (**BALL**)	22
	Toroid cutter (**TORUS**)	23
	Chamfer mill (**MILL_CHAMFER**)	24
	Side milling cutter (**MILL_SIDE**)	25
	Drill (**DRILL**)	1
	Tap (**TAP**)	2
	NC center drill (**CENT**)	4
	CAL_PIN	28
	Touch probe (**TCHP**) (#17 / #1-05-1)	21
	Reamer (**REAM**)	3
	Countersink (**CSINK**)	5
	Piloted counterbore (**TSINK**)	6
	Boring tool (**BOR**)	7

Icon	Tool type	Number
	Back boring tool (**BCKBOR**)	8
	Thread miller (**GF**)	15
	Thread miller with chamfer (**GSF**)	16
	Thread mill with single thread (**EP**)	17
	Thread mill with indexable insert (**WSP**)	18
	Thread drilling/milling cutter (**BGF**)	19
	Circular thread milling drill (**ZBGF**)	20

8.4 Tool management

Application

The control displays the tool definitions of all technologies as well as the tools currently present in the tool magazine in the **Tool management** application of the **Tables** operating mode.

The tool management allows you to add or delete tools, or to edit parameters.

Related topics

■ Creating new tools

Further information: "Setting up a tool", Page 83

■ Importing and exporting tool data (e.g., from a tool presetter or for the programming station)

Further information: "Importing and exporting of table contents", Page 273

■ Table workspace

Further information: "The Table workspace", Page 262

■ Form workspace

Further information: "The Form workspace for tables", Page 270

Description of function

You can define up to 32,767 tools in the tool management; this is the maximum number of available table rows.

In the tool management, the control displays the parameters of the following tool tables:

■ Tool table **tool.t**

Further information: "Tool table tool.t", Page 278

■ Touch-probe table **tchprobe.tp** (#17 / #1-05-1)

Further information: "Touch probe table tchprobe.tp (#17 / #1-05-1)", Page 285

In the tool management, the control also displays parameters from the **tool_p.tch** pocket table (e.g., **P**).

Further information: "Pocket table tool_p.tch", Page 290

You can edit the parameters in the **Table** or **Form** workspaces. In the **Form** workspace the control shows the correct parameters for each tool type.

Further information: "Tool parameters", Page 154

Notes

- When creating a new tool, the length **L** and radius **R** parameters are empty at first. The control will not insert a tool whose length and radius are missing and will display an error message.

- If a tool has been saved to the pocket table, you can neither reset the corresponding row in tool management nor delete the tool. The tool must be removed from the magazine first.

- When editing parameters, bear in mind that the current tool may have been entered in the **RT** column as a replacement tool of another tool!

- Make sure to keep the tool table as short and clear as possible so that it does not impair the computing speed of your control. Use a maximum of 10,000 tool entries in tool management. For example, you can delete all unused tool numbers; tool numbers need not be sequential.

- In the **Form** workspace, the control displays the relevant parameters for the current tool type only and hides all other parameters. If you select the **Configure the layout** icon, the control will also display the hidden form areas.
  Further information: "Adding a column in the workspace", Page 272

- If the cursor is within the **Table** workspace and the **Edit** toggle switch is deactivated, a search using the keyboard can be started. The control opens a separate window with an input field and automatically searches for the entered string. If it finds a tool with the entered characters, the control selects this tool. If it finds several tools with this string of characters, you can scroll up and down in the window

- In the **Simulation** workspace you can check for collisions between the tool (including its holder) and the workpiece or fixtures.

- With the optional machine parameter **resetOnTypeChange** (no. 125304), you can define how the control will react when the tool type is changed. The machine manufacturer enables this parameter. If this machine parameter has been set to **TRUE** and you change the tool type, the control will first display a confirmation prompt and then reset all tool parameters.

- Changing a tool parameter in the pocket table also changes that same parameter in the tool manager.

- In the **Table** workspace of the tool manager, the control displays the additional virtual **MAGAZINE** and **TOOL_LIFE** columns.
  Further information: "Virtual columns", Page 269

8.5 Tool carrier management

Application

With tool carrier management, you can assign the 3D model of a tool carrier to a tool.

The tool carrier model will be used for the following functions:

■ Representation in the **Simulation** workspace

Related topics

■ The **Simulation** workspace
Further information: Programming and Testing User's Manual

■ Validating a 3D model for the tool carrier (#56-61 / #3-02-1*)
Further information: "OPC UA NC Server (#56-61 / #3-02-1*)", Page 362

Requirements

■ Kinematics description
The machine manufacturer creates the kinematics description.

■ The insertion point is defined within the machine's kinematic description.
The machine manufacturer defines the insertion point for the tool carrier.

■ Tool carrier model exists
You must save the tool carrier model in the **Toolkinematics** folder.
Path: **TNC:\system\Toolkinematics**

■ The tool carrier model has been assigned to the tool
Further information: "Assigning a tool carrier", Page 166

Description of function

The tool carrier model must meet the following requirements:

■ Use permitted characters for the file name
Further information: Programming and Testing User's Manual

■ Use a supported format
  ■ CFG file
  ■ M3D file
  ■ STL file
    ■ Max. 20 000 triangles
    ■ Triangular mesh forms a closed shell
      Further information: "Generating STL files with 3D mesh (#152 / #1-04-1)", Page 193

If you are using CFT or CFX files, you must edit the templates in the **ToolHolderWizard** window.

Further information: "Customizing tool carrier templates with ToolHolderWizard", Page 167

8.5.1 Assigning a tool carrier

To assign a tool carrier to a tool:

▶ Select the **Tables** operating mode

▶ Select the **Tool management** application

▶ Select the tool you want to use

▶ Enable **Edit**

▶ If applicable, open the **Form** workspace

▶ In the **Additional geometry data** area, select the **KINEMATIC** parameter

> The control displays the available tool carriers in the **Tool-carrier kinematics** window.
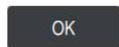
▶ Select the desired tool carrier

▶ Select **OK**

> The control assigns the 3D model of the tool carrier to the tool.

> ⓘ The tool carrier will be taken into account only after the next tool call.

Notes

■ Sample files for tool carrier templates are available on the programming station in the **TNC:\system\Toolkinematics** folder.

■ In the **Simulation** workspace you can check for collisions between the tool (including its holder) and the workpiece or fixtures.

■ On 3-axis machines with rectangular angle heads, tool carriers of angle heads are advantageous in connection with the tool axes **X** and **Y** because the control takes the dimensions of the angle heads into account.

■ Even if the inch unit of measure is active in the control or NC program, the control will interpret dimensions of 3D files in mm.

8.6 Customizing tool carrier templates with ToolHolderWizard

> **i** Many tool carriers only differ from others in terms of their dimensions, but their geometric shape is identical. HEIDENHAIN provides ready-to-use tool carrier templates for downloading. Tool carrier templates are 3D models with fixed geometries but editable dimensions.
>
> They can be downloaded through the following link:
>
> **HEIDENHAIN NC solutions**
>
> If you need further tool carrier templates, please contact your machine manufacturer or third-party vendor.

If you would like to use a CFX or CFT file, you need to parameterize the tool carrier template (i.e., to define the required dimensions). The tool carrier templates can be parametrized in the **ToolHolderWizard** window.

The **ToolHolderWizard** window contains the following icons:

Icon	Meaning
	Close the application
	Open file
	Switch between wire frame model and solid object view
	Switch between shaded and transparent view
	Show or hide **Transformation vectors**
	Show or hide **Names of collision objects**
	Show or hide **Test points**
	Show or hide **Measuring points**
	Redo (restore) the initial view
	Orientations (e.g., top view)

8.6.1 Parameterizing tool carrier templates

To parameterize a tool carrier template:

▶ Select the **Files** operating mode

▶ Open the **TNC:\system\Toolkinematics** folder
▶ Double-tap or double-click desired tool carrier template with the ***.cft** extension
> The control opens the **ToolHolderWizard** window.
▶ Define the dimensions in the **Parameter** area
▶ Define a name with the ***.cfx** extension in the **Output file** area
▶ Select **Generate file**
> The control shows the message that the tool carrier template was successfully generated and saves the file in the folder **TNC:\system\Toolkinematics**.
▶ Select **OK**
▶ Select **Close the application**

ⓘ Parameterized tool carriers can consist of several subfiles. If the subfiles are incomplete, the control will display an error message.

Only use fully parameterized tool carriers and error-free STL or M3D files!

8.7 Tool usage test

Application

The tool usage test allows checking the tools used in the NC program before starting the program. The control checks if the tools used are available in the machine magazine and have sufficient remaining tool life. Any missing tools can be stored in the machine or tools can be exchanged due to insufficient remaining tool life before starting the program. This avoids interruptions while the program is running.

Related topics

■ Contents of the tool usage file

  Further information: "Tool usage file", Page 293

Requirements

■ To perform a tool usage test, you need a tool usage file

  In the machine parameter **createUsageFile** (no. 118701), the machine manufacturer defines whether the **Generate tool-usage file** function will be enabled.

  Further information: "Tool usage file", Page 293

■ The **Generate tool-usage file** setting is set to **Once** or **Always**

  Further information: "The Machine and Simulation areas", Page 333

■ Use the same tool table for the simulation as for program run

  Further information: Programming and Testing User's Manual

Description of function

Creating the tool usage file

A tool usage file must be generated for performing the tool usage test.

When setting the **Generate tool-usage file** setting to **once** or **always**, the control will generate a tool usage file in the following cases:

■ Simulating the NC program completely

■ Executing the NC program completely

■ Select the **Refresh** icon in the **Tool usage** area of the **Tests** column

  Further information: Programming and Testing User's Manual

The control saves the tool usage file with the ***.t.dep** extension in the same folder where the NC program is stored.

Further information: "Tool usage file", Page 293

8.7.1 Performing the tool usage test

To perform a tool usage test:
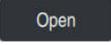
🖹	▶ Select the **Editor** operating mode
✚	▶ Select **Add**
	▶ Select the desired NC program
Open	▶ Select **Open**
	> The control opens the NC program in a new tab.
⊘	▶ Open the **Tests** column
↻	▶ In the **Tool usage** area, select **Refresh**
	> The control generates a tool usage file and displays the tools used in the **Tool usage** area.
	Further information: "Tool usage file", Page 293
↻	▶ In the **Tool check** area, select **Refresh**
	> The control performs the tool usage test.
	> The **Tool check** area shows whether all tools are available and have sufficient remaining tool life.
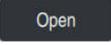
Notes

- If you double-tap or double-click a tool entry in the **Tool usage** or **Tool check** areas, the control switches to the tool selected in tool management. You can make modifications as needed.
- The **Simulation settings** window allows selecting when the control generates a tool usage file for the simulation.
  Further information: Programming and Testing User's Manual
- The control saves the tool usage file as a dependent file (***.dep**).
  Further information: "Tool usage file", Page 293
- In the settings of the **Files** operating mode, you can specify whether the control displays dependent files in the file management.
  Further information: Programming and Testing User's Manual
- The function **FN 18: SYSREAD ID975 NR1** allows querying the tool usage test for an NC program.
- The machine manufacturer uses the machine parameter **autoCheckPrg** (no. 129801) to define whether the control automatically generates a tool usage file upon selecting an NC program.

9

Coordinate transformation

9.1 Reference systems

9.1.1 Overview

A control requires unambiguous coordinates in order to move an axis to a defined position correctly. For coordinates to be unambiguous, they not only require the values but also a reference system in which these values are valid.

The control differentiates between the following reference systems:

Abbrevia-tion	Meaning	Further information
M-CS	Machine coordinate system machine coordinate system	Page 174
B-CS	Basic coordinate system basic coordinate system	Page 175
W-CS	Workpiece coordinate system workpiece coordinate system	Page 176
I-CS	Input coordinate system input coordinate system	Page 177
T-CS	Tool coordinate system tool coordinate system	Page 178

The control uses different reference systems for different purposes. For example, this makes it possible to always exchange tools at the exact same position while maintaining the possibility of adapting an NC program to the workpiece position.

The reference systems build upon each other. The machine coordinate system **M-CS** is the fundamental reference system. The positions of the following reference systems are determined by transformations from this basis.

Definition
Transformations
Translatory transformations each enable a shift along a number line.

9.1.2 Basics of coordinate systems

Types of coordinate systems

For coordinates to be unambiguous they must define one point in all axes of the coordinate system:

Axes	Function
One	In a one-dimensional coordinate system, one coordinate defines one point on a number line.
	Example: on a machine tool, a linear encoder represents a number line.
Two	In a two-dimensional coordinate system, two coordinates define one point in a plane.
Three	In a three-dimensional coordinate system, three coordinates define one point in space.

If the axes are arranged perpendicularly to each other, they create a Cartesian coordinate system.

Using the right-hand rule you can recreate a three-dimensional Cartesian coordinate system. The fingertips point in the positive directions of the three axes.
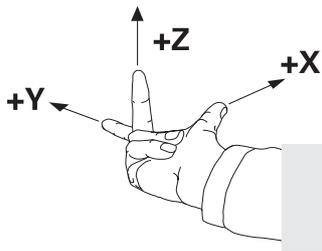
Origin of the coordinate system

Unambiguous coordinates require a defined reference point to which the values refer, starting from zero. This point is the coordinate origin, which lies at the intersection of the axes for all three-dimensional Cartesian coordinate systems of the control. The coordinate origin has the coordinates **X+0**, **Y+0**, and **Z+0**.
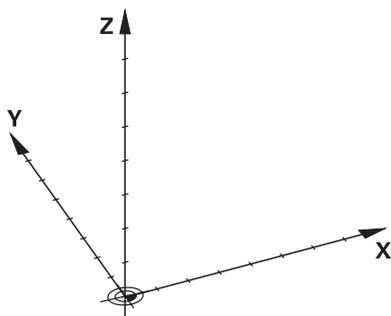
9.1.3 Machine coordinate system M-CS

Application

In the machine coordinate system **M-CS** you program constant positions, such as a safe position for retraction. The machine manufacturer also defines constant positions in the **M-CS**, such as the tool-change point.

Description of function

Properties of M-CS machine coordinate system

The machine coordinate system **M-CS** corresponds to the kinematics description and therefore to the actual mechanical design of the machine tool. The physical axes of a machine tool are not necessarily always exactly perpendicular to each other, and therefore do not represent a Cartesian coordinate system. The **M-CS** thus consists of multiple one-dimensional coordinate systems that correspond to the axes of the machine.

The machine manufacturer defines the position and orientation of the one-dimensional coordinate systems in the kinematics description.
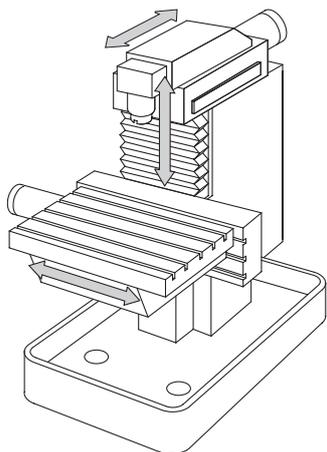
The machine datum is the coordinate origin of the **M-CS**. The machine manufacturer defines the machine datum in the machine configuration.

The values in the machine configuration define the zero positions of the position 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 also be located outside of the traverse range.

Position display

The following modes of the position display are referenced to the machine coordinate system **M-CS**:

- **Nominal reference position (RFNOML)**
- **Actual reference position (RFACTL)**
- **Nominal distance to go (REFDST)**

The difference between the values of the **RFACTL** and **ACTL.** modes of an axis results from all active transformations in other reference systems.

Programming coordinate entry in machine coordinate system M-CS

With miscellaneous function **M91** you program the coordinates relative to the machine datum.

Further information: Programming and Testing User's Manual

9.1.4 Basic coordinate system B-CS

Application

In the basic coordinate system **B-CS** you define the position of the workpiece. You determine these values by using a 3D touch probe, for example. The control saves the values in the preset table.

Description of function

Properties of the basic coordinate system B-CS

The basic coordinate system **B-CS** is a three-dimensional Cartesian coordinate system. Its coordinate origin is the end of the kinematics description.

The machine manufacturer defines the coordinate origin and orientation of the **B-CS**.

Transformations in the basic coordinate system B-CS

The control saves the determined values as basic transformations in the **B-CS** in the preset table.
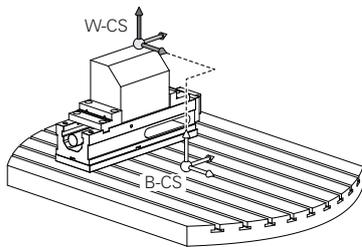
Further information:

9.1.5 Workpiece coordinate system W-CS

Application

In the workpiece coordinate system **W-CS** you define the position of the working plane. Program the transformations for this purpose.

Description of function

Properties of the workpiece coordinate system W-CS

The workpiece coordinate system **W-CS** is a three-dimensional Cartesian coordinate system. Its coordinate origin is the active workpiece preset from the preset table.

Further information: "Preset management", Page 179
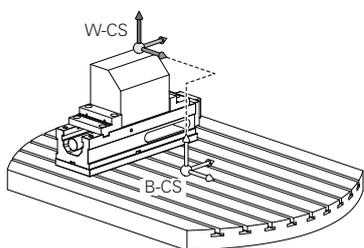
Transformations in the workpiece coordinate system (W-CS)

HEIDENHAIN recommends using the following transformations in the **W-CS** workpiece coordinate system:

- Axes **X**, **Y**, **Z** of the **TRANS DATUM** function
- The **TRANS MIRROR** function or cycle **8 MIRRORING**
- The **TRANS SCALE** function or cycle **11 SCALING FACTOR**
- Cycle **26 AXIS-SPECIFIC SCALING**

Further information: Programming and Testing User's Manual

With these transformations you modify the position of the **I-CS** input coordinate system.

Note

If you do not define any transformation, the positions of the **W-CS** and the **I-CS** are identical.

9.1.6 Input coordinate system I-CS

Application

The programmed values in the NC program refer to the input coordinate system **I-CS**. You use positioning blocks to program the position of the tool.

Description of function

Properties of the input coordinate system I-CS

The input coordinate system **I-CS** is a three-dimensional Cartesian coordinate system. You define the coordinate origin of the **I-CS** by means of transformations in the workpiece coordinate system **W-CS**.

Further information: "Workpiece coordinate system W-CS", Page 176

If you do not define any transformation, the positions of the **W-CS** and the **I-CS** are identical.
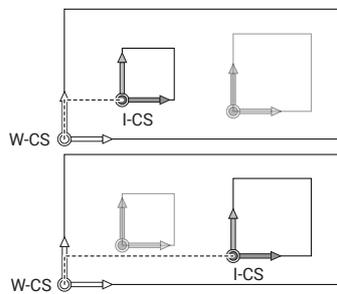
Position display

The following modes of the position display are referenced to the input coordinate system **I-CS**:

- **Nominal pos. (NOML)**
- **Actual pos. (ACT)**
- **Actual distance to go (ACTDST)**

9.1.7 Tool coordinate system T-CS

Application

The control implements tool compensations in the tool coordinate system **T-CS**.

Description of function

Properties of the tool coordinate system T-CS

The tool coordinate system **T-CS** is a three-dimensional Cartesian coordinate system. Its coordinate origin is the tool tip TIP.

By making entries in the tool management, you can define the tool tip relative to the tool carrier reference point. The machine manufacturer usually defines the tool carrier reference point on the spindle tip.

Further information: "Presets in the machine", Page 148

Use the following tool management parameters to define the tool tip relative to the tool carrier reference point:

- **L**
- **DL**

Further information: "Tool carrier reference point", Page 151

You can use positioning blocks in the input coordinate system **I-CS** to define the position of the tool and therefore the position of the **T-CS**.

Further information: "Input coordinate system I-CS", Page 177

You can use miscellaneous functions to also program in other reference systems, such as **M91** for the machine coordinate system **M-CS**.

Further information: Programming and Testing User's Manual

Transformations in the tool coordinate system T-CS

The following tool compensations have an effect in the tool coordinate system **T-CS**:

- Compensation values from the tool management
- Compensation values from the tool call

  Further information: Programming and Testing User's Manual

9.2 Preset management

Application

The preset management allows setting and activating single presets. The presets to be saved may include, for example, the position of a workpiece in the preset table. The active row in the preset table is used as a workpiece preset in the NC program and as the coordinate origin of the workpiece coordinate system **W-CS**.

Further information: "Presets in the machine", Page 148

Use the preset management if you have been working with previous control models using REF-based datum tables.

Related topics

- Contents of preset table, write protection

  Further information: "Preset table *.pr", Page 295

Description of function

Setting presets

Presets can be set in the following ways:

- Setting axis positions manually

  Further information: "Setting a preset manually", Page 181
- Touch probe cycles in the **Setup** application

  Further information: "Touch probe functions in the Manual operating mode (#17 / #1-05-1)", Page 209
- Touch probe cycles in the NC program (#17 / #1-05-1)

If you try to write a value in a write-protected preset table row, the control cancels this process with an error message. Write-protection for this row must be rescinded first.

Further information: "Removing write protection", Page 298

Setting a preset with milling cutters

If no workpiece touch probe is available, the preset can also be set by using a milling cutter. In this case, the values are not obtained by probing, but by scratching.
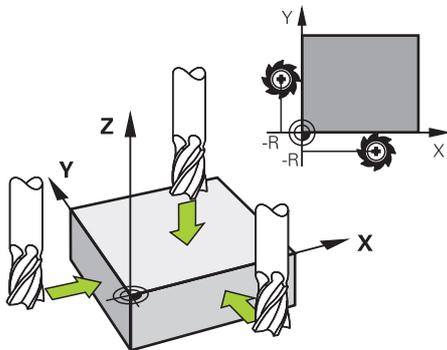
When scratching with a milling cutter, the tool is slowly moved to the workpiece edge in the **Manual operation** application while the spindle is rotating.

As soon as the tool produces chips on the workpiece, set the preset manually in the desired axis.

Further information: "Setting a preset manually", Page 181

Activating presets

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. If the previous value is kept, there is a danger of collision!

▶ Before activating a preset, check whether all columns contain values.

▶ For undefined columns, enter values (e.g., **0**)

▶ As an alternative, have the machine manufacturer define **0** as the default value for the columns

---

Presets can be activated in the following ways:

■ Activating manually in the **Tables** operating mode
Further information: "Activating a preset manually", Page 182

■ Cycle **247 PRESETTING**
Further information: Programming and Testing User's Manual

■ **PRESET SELECT** function
Further information: Programming and Testing User's Manual

When activating a preset, the control resets the following transformations:

■ Datum shift with the **TRANS DATUM** function

■ Mirroring with the **TRANS MIRROR** function or Cycle **8 MIRRORING**

■ Scaling with the **TRANS SCALE** function or cycle **11 SCALING FACTOR**

■ Axis-specific scaling with Cycle **26 AXIS-SPECIFIC SCALING**

9.2.1 Setting a preset manually
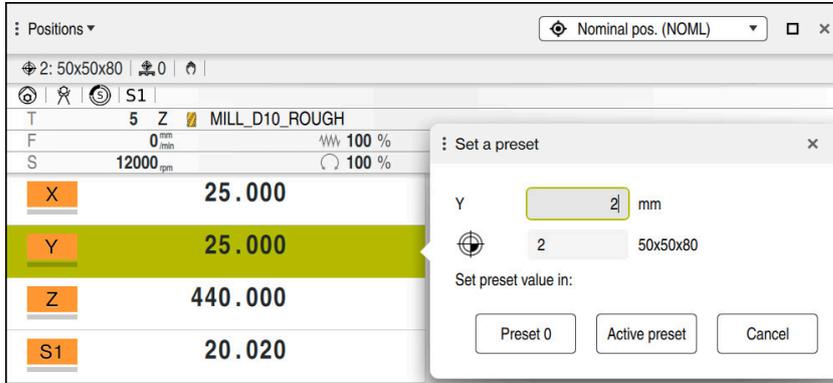
The **Set a preset** window in the **Positions** workspace

When setting the preset manually, the values can be written either in row 0 of the preset table or in the active row.

To set a preset manually in an axis:

▶ Select the **Manual operation** application in the **Manual** operating mode

▶ Open the **Positions** workspace

▶ Traverse the tool to the desired position (e.g., for scratching)

▶ Select the row of the desired axis

▶ Select **Set the preset**

> The control opens the **Set a preset** window.

▶ Enter the value of the current axis position, relating to the new preset (e.g., **0**)

> The control activates the **Preset 0** and **Active preset** buttons for selection.

▶ Select an option (e.g., **Active preset**)

> The control saves the value in the selected preset table row and closes the **Set a preset** window.

> The control updates the values in the **Positions** workspace.

ℹ
- When selecting **Preset 0**, the control automatically activates row 0 of the preset table as the workpiece preset.
- Use the **+**, **-**, *****, **/**, **(**, and **)** keys for calculations in the numerical input fields.

9.2.2   Activating a preset manually

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. If the previous value is kept, there is a danger of collision!

► Before activating a preset, check whether all columns contain values.
► For undefined columns, enter values (e.g., **0**)
► As an alternative, have the machine manufacturer define **0** as the default value for the columns

---

To activate a preset manually:

⊞	► Select the **Tables** operating mode
⊕	► Select the **Presets** application
	► Select the desired row
Activate the preset	► Select **Activate the preset**
	> The control activates the preset.
	> The control displays the number and comment of the active preset in the **Positions** workspace and in the status overview.

Further information: "Description of function", Page 95
Further information: "Status overview of the TNC bar", Page 99

Notes

- In the optional machine parameter **initial** (no. 105603), the machine manufacturer defines a default value for every column of a new row.
- In the optional machine parameter **CfgPresetSettings** (no. 204600), the machine manufacturer can block the setting of a preset in individual axes.
- When scratching a workpiece with the radius of a milling cutter, the radius value must be taken into account in the preset.

10

CAD Viewer

10.1   Fundamentals

Application

CAD Viewer allows you to open CAD files on the control (e.g., to set a workpiece preset in the model and load it to an NC program).

The **3D mesh** (#152 / #1-04-1) function allows you to rectify errors in 3D models.

Related topics

- Rectifying errors in 3D models with **3D mesh** (#152 / #1-04-1)

  Further information: "Generating STL files with 3D mesh (#152 / #1-04-1)", Page 193

Description of function

CAD Viewer runs as a separate application on the third desktop of the control.

CAD Viewer supports the following standard file types that can be opened directly in the control:

File type	Extension	Format
STEP	*.stp and *.step	■ AP 203 ■ AP 214
IGES	*.igs and *.iges	■ Version 5.3
DXF	*.dxf	■ R10 to 2015 ■ ASCII
STL	*.stl	■ Binary ■ ASCII

Screen layout
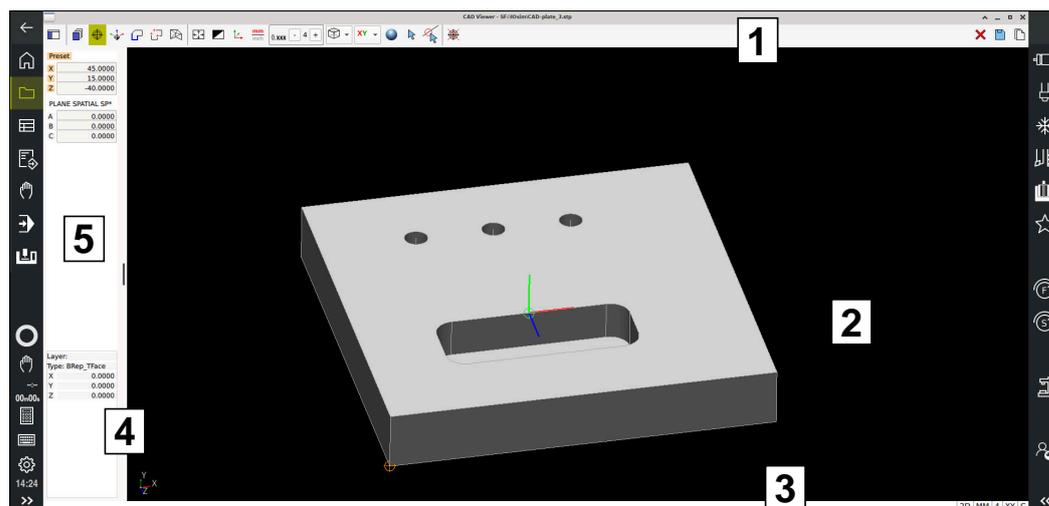
CAD file open in **CAD Viewer**

CAD Viewer contains the following areas:

1   Menu bar

    Further information: "Menu bar icons", Page 185

2   Graphics area

    In the Graphics area, the control displays the CAD model.

3   Status bar

In the status bar, the control displays the active settings.

4   Element information area

Further information: "The Element Information area", Page 187

5   List view area

The List View area displays information on the active function (e.g., available layers or the position of the workpiece preset).

Menu bar icons

The menu bar contains the following icons:

Icon	Meaning
	Show sidebar Show, enlarge or hide the List View and Element Information areas
	Display the layer Display the layer(s) in the List View area **Further information:** "Layer", Page 187
	Preset Define the workpiece preset
	Workpiece preset has been defined
	Delete the defined workpiece preset **Further information:** "Workpiece preset in the CAD file", Page 189
	Datum Set the datum
	Datum has been set **Further information:** "Workpiece datum in the CAD file", Page 192
	Contour Without function for TNC7 go
	Positions Without function for TNC7 go
	3D mesh Create a 3D mesh (#152 / #1-04-1) **Further information:** "Generating STL files with 3D mesh (#152 / #1-04-1)", Page 193
	Show all Set the zoom to the largest possible view of the complete graphics
	Inverted colors Change the background color (black or white)
	Toggle between 2D and 3D modes

Icon	Meaning
mm/inch	Set the unit of measure (mm or inches) **CAD Viewer** always calculates internally with mm. If you select inches as the unit of measure, **CAD Viewer** will convert all values to inches.
0.01 ... 0.001	**Number of decimal places** Select decimal places: **3...7** for mm and **4...8** for inches The decimal places define the resolution and the number of segments during linearization. Default setting: 4 decimal places with **mm**, and 5 decimal places with **inch** as the unit of measure
(cube icon)	**Set perspective** Switch between various views of the model (e.g., **Top**)
XY	**Axes** Select the working plane: ▪ **XY** ▪ **YZ** ▪ **ZX** ▪ **ZXØ**
(sphere icon)	Toggle a 3D model between a solid model and a wire-frame model.
(cursor icon)	**Select** contour elements (e.g., to obtain element information) **Further information:** "The Element Information area", Page 187
+	**Add** contour elements
−	**Remove** contour elements The icon shows the current mode. Clicking the icon activates the next mode.
(cursor with line icon)	Activate or deactivate **Hover** If you swipe across the CAD model during touch operation, the controls display selectable options in orange. The function is comparable to hovering with a mouse. As soon as you stop the movement, the control will mark the current selection option. If this icon is active, you can no longer rotate the CAD model.
(undo icon)	**Undo an action**
✗	**Delete entire list**
(save icon)	**Save entire list content to a file**
(copy icon)	**Copy entire list contents to clipboard** The control retains the content of the clipboard only as long as **CAD Viewer** is open.

The Element Information area

In the Element Information area, the following information is displayed for the selected element of the CAD file:

■ Associated layer

■ Element type

■ Point type:

   ■ Point coordinates

■ Line type:

   ■ Coordinates of the starting point

   ■ Coordinates of the end point

■ Circular arc or circle type:

   ■ Coordinates of the starting point

   ■ Coordinates of the end point

   ■ Coordinates of the center point

   ■ Radius

The control always shows the **X**, **Y** and **Z** coordinates. In 2D mode, the Z coordinate is dimmed.

Layer

CAD files usually contain multiple layers. The designer uses these layers to create groups of various types of elements, such as the actual workpiece contour, dimensions, auxiliary and design lines, hatching, and texts.

The CAD file to be processed must contain at least one layer. The control automatically moves all elements not assigned to a layer to the "anonymous" layer.

If the name of the layer is not shown completely in the List View area, you can use the **Show sidebar** icon to enlarge this area.

Use the **Display the layer** icon to display all the layers of the file in the List View area. Use the check box in front of the name to show and hide individual layers.

When you open a CAD file in **CAD Viewer**, all available layers are shown.

If you hide unnecessary layers, the graphic becomes clearer.

Manipulating the graphics in CAD Viewer

You can use the mouse or touch gestures to perform the following functions on the CAD model:

Function	Mouse operation	Touch operation	
Show the selection options on the CAD model in orange	Hover over the element with the mouse pointer	Swipe across the elements	
		The **Hover** mode is active	
		ⓘ As soon as you stop the movement, the control will mark the current selection option.	
Zooming Magnify or reduce the CAD model	Scroll with the mouse wheel	Spread or pinch	
Move the CAD model	Drag with the right mouse button held down	Two-finger drag	
Reset the CAD model to its original size	Double-click with the right mouse button	Double tap	
Reset the CAD model to its original size and angle The **Add** mode	Double-click with the right mouse button Press the **SHIFT** key in other modes	Double tap	
Rotate the CAD model The **Hover** mode is inactive	Drag with the left mouse button held down	Swipe	

Notes

- Before loading the file into the control, ensure that the name of the file contains only permitted characters.

  Further information: Programming and Testing User's Manual
- When you select a layer in the List View area, you can press the spacebar to show and hide the layer.
- **CAD Viewer** allows you to open CAD files consisting of any number of triangles.

10.2 Workpiece preset in the CAD file

Application

The datum of the drawing in the CAD file is not always located in a manner that lets you use it as a workpiece preset. Therefore, the control provides a function with which you can shift the workpiece preset to a suitable location by clicking an element. You can also define the orientation of the coordinate system.

Related topics

- Presets in the machine

  Further information: "Presets in the machine", Page 148

Description of function

When you select the **Preset** icon, the control displays the following information in the list view area:

- Distance between the defined preset and the drawing datum
- Orientation of the working plane

The control displays values not equal to 0 in orange.
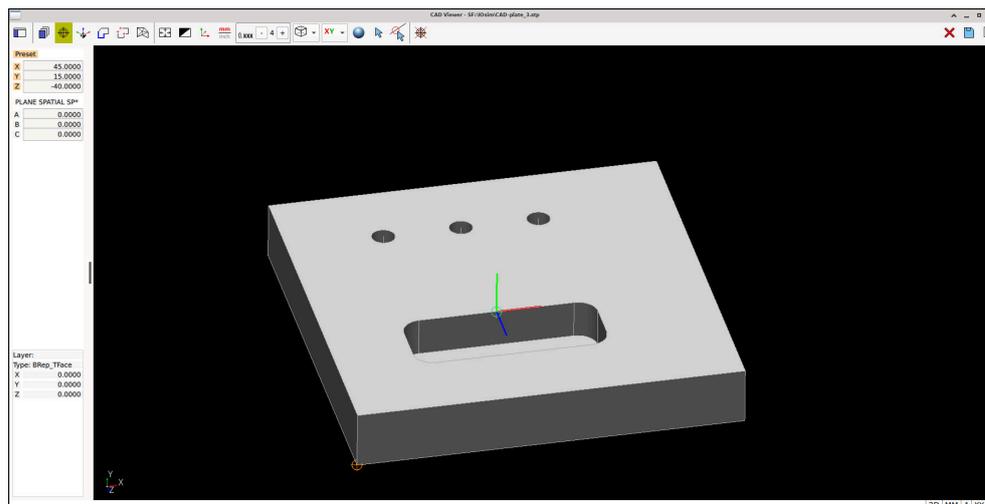
Workpiece preset in the CAD file

You can position the preset at the following locations:

- By direct input of numerical values into the List View area
- For lines:
    - Starting point
    - Midpoint
    - End point
- For circular arcs:
    - Starting point
    - Midpoint
    - End point
- For full circles:
    - At the quadrant transitions
    - At the center

- At the intersection between:
    - Two lines, even if the point of intersection is actually on the extension of one of the lines
    - Line and circular arc
    - Line and full circle
    - Two circles (regardless of whether a circular arc or a full circle)

If you have set a workpiece preset, the control displays the **Preset** icon in the menu bar with a yellow quadrant.

You can save the workpiece preset and workpiece datum information to a file or the clipboard even without the CAD Import software option.

> **i** The control retains the content of the clipboard only as long as **CAD Viewer** is open.

10.2.1 Setting the workpiece preset or workpiece datum and orienting the coordinate system

> **i**
> - The following instructions also apply to the workpiece datum. In this case, start by selecting the **Datum** icon.
> - The following instructions apply to the use of a mouse. You can also perform all steps with touch gestures.
>   **Further information:** "Manipulating the graphics in CAD Viewer", Page 188

Setting the workpiece preset or workpiece datum on an individual element

To set the workpiece preset on an individual element:

- ▶ Select **Preset**
- ▶ Position the cursor on the desired element
- > If you are using a mouse, the control displays selectable presets for the element using gray icons.
- ▶ Click the icon at the desired position
- > The control sets the workpiece preset to the selected position. The control turns the icon green.
- ▶ Orient the working plane, if required

Setting the workpiece preset or workpiece datum at the intersection of two elements

You can set the workpiece preset at the intersection of lines, full circles, and arcs.

To set the workpiece preset at the intersection of two elements:

▶ Select **Preset**

▶ Click the first element
> The control highlights the element in color.
▶ Click the second element
> The control sets the workpiece preset at the point of intersection of the two elements. The control marks the workpiece preset with a green symbol.
▶ Orient the working plane, if required

> ℹ | ▪ If there are several possible intersections, the control selects the intersection nearest the mouse-click on the second element.
> | ▪ If two elements do not intersect directly, the control automatically calculates the intersection of their extensions.
> | ▪ If the control cannot calculate an intersection, it deselects the previously selected element.

Orienting the working plane

The following requirements must be met in order to orient the working plane:
▪ Preset has been defined
▪ There are elements next to the preset that can be used for the desired orientation

To orient the working plane:
▶ Select an element in the positive direction of the X axis
> The control orients the X axis.
> The control changes the **C** angle in the List View area.
▶ Select an element in the positive direction of the Y axis
> The control orients the Y and Z axes.
> The control changes the **A** and **C** angles in the List View area.

10.3   Workpiece datum in the CAD file

Application

The workpiece preset is not always located in a manner that lets you machine the entire part. Therefore, the control has a function with which you can define a new datum and a working plane.

Related topics

- Presets in the machine

  Further information: "Presets in the machine", Page 148

Description of function

When you select the **Datum** icon, the control displays the following information in the list view area:

- Distance between the datum that has been set and the workpiece preset
- Orientation of the working plane

You can apply a workpiece datum set in CAD Viewer and shift it, if required, by entering values directly in the List View area.

The control displays values not equal to 0 in orange.

The datum with the orientation of the working plane can be set at the same positions as a preset.

Further information: "Workpiece preset in the CAD file", Page 189

If you have set a workpiece datum, the control displays the **Datum** icon in the menu bar with a yellow area.

Further information: "Setting the workpiece preset or workpiece datum and orienting the coordinate system", Page 190

You can save the workpiece preset and workpiece datum information to a file or the clipboard even without the CAD Import software option.

> **ℹ**   The control retains the content of the clipboard only as long as **CAD Viewer** is open.

10.4 Generating STL files with 3D mesh (#152 / #1-04-1)

Application

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

Related topics

- Exporting the simulated workpiece as an STL file

Requirement

- Software option CAD Model Optimizer (#152 / #1-04-1)

Description of function

When you select the **3D mesh** icon, the control changes to **3D mesh** mode. The control covers the 3D model displayed in **CAD Viewer** with a mesh of triangles.

The control simplifies the original model and removes errors, such as small holes in a solid or self-intersections of a surface.

You can save the result and use it for various control functions, for example as a workpiece blank with the **BLK FORM FILE** function.
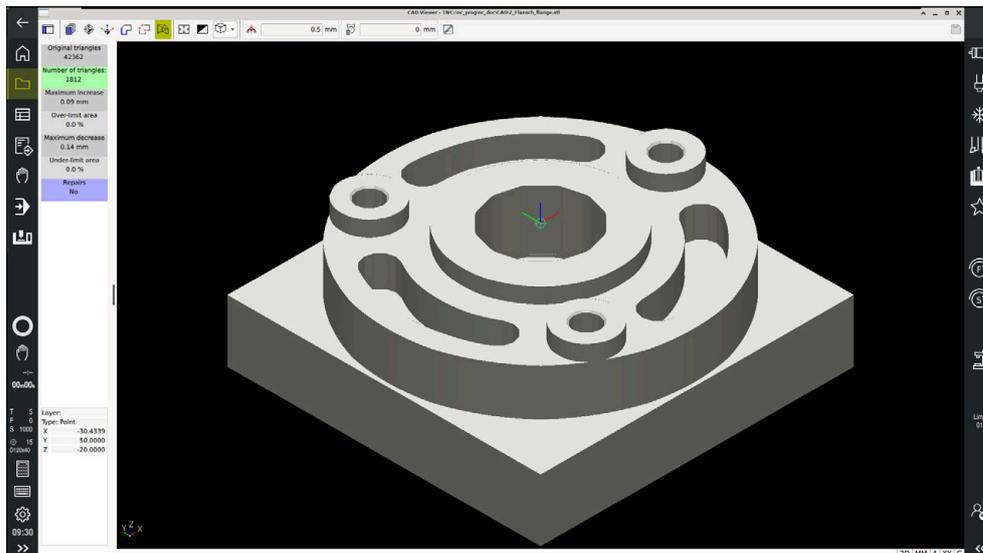
3D model in **3D mesh** mode

The simplified model or parts of it may be smaller or larger than the original model. The result depends on the quality of the original model and the settings selected in **3D mesh** mode.

The List View area shows the following information:

Option	Meaning
Original triangles	Number of triangles in the original model
Number of triangles:	Number of triangles with active settings in the simplified model

> ℹ️ If this option is highlighted in green, the number of triangles is in the optimum range.
>
> You can further reduce the number of triangles using the available functions.
>
> **Further information:** "Functions for the simplified model", Page 195

Option	Meaning
Maximum increase	Maximum increase of the triangle mesh
Over-limit area	Surface increase in percent compared to the original model
Maximum decrease	Maximum decrease of the triangle mesh compared to the original model
Under-limit area	Surface decrease in percent compared to the original model
Repairs	Indicates whether the original model has been repaired or not
	If it has been repaired, the control indicates the type of repair (e.g., **Hole Int Shells**).
	This indication consists of the following items:
	▪ **Hole**
	CAD Viewer closed holes in the 3D model.
	▪ **Int**
	CAD Viewer removed self-intersections.
	▪ **Shells**
	CAD Viewer joined multiple separate solids.
Time	Duration of simplification

In order to use STL files for control functions, the saved files must meet the following requirements:

▪ Max. 20 000 triangles
▪ Triangular mesh forms a closed shell

The greater the number of triangles in an STL file, the greater the processing power required by the control for simulation.

Functions for the simplified model

In order to reduce the number of triangles, you can define further settings for the simplified model.

CAD Viewer provides the following functions:

Icon	Meaning
	Allowed simplification
	Use this function to simplify the output model by the specified tolerance. The higher the value, the more the surfaces may deviate from the original.
	Remove holes <= diameter
	Use this function to remove holes and pockets up to the specified diameter from the original model.
	Only optimized mesh shown
	The control shows the simplified model only.
	Original is displayed
	The control shows the simplified model, superimposed with the original mesh from the original file. You can use this function to evaluate deviations.
	Save
	Use this function to save the simplified 3D model with the selected settings as an STL file.

11

User aids

11.1 Virtual keyboard of the control bar

Application

You can use the virtual keyboard for entering NC functions, letters, and numbers, and for navigation.

The virtual keyboard offers the following modes:

- NC input
- Text input
- Formula entry

Description of function

The control opens NC input mode by default after the start procedure.

You can move the keyboard on the screen. The keyboard remains active, even when the operating mode is switched, until the keyboard is closed.

The control remembers the position and mode of the virtual keyboard until it is shut down.

The **Keyboard** workspace provides the same functions as the virtual keyboard.

The **+**, **-**, *****, **/**, **(** and **)** keys permit calculations concerning numerical values in input fields and table rows.

NC input areas
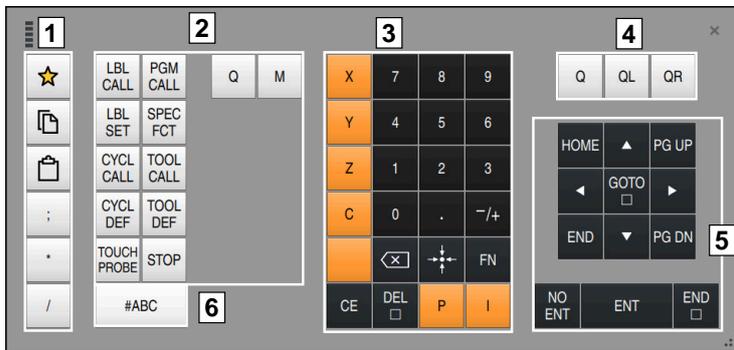
Virtual keyboard in NC input mode

NC input mode contains the following areas:

1   File functions
- Define favorites
- Copy
- Paste
- Add comment
- Add structure item
- Hide NC block
2   NC functions
3   Axis keys and numerical input
4   Q parameters
5   Navigation and dialog keys
6   Switch to text input

> 🛈 If you press the **Q** button in the NC functions area repeatedly, the control cycles through the syntax in the following sequence:
> - **Q**
> - **QL**
> - **QR**

Text input areas

Virtual keyboard in text input mode

Text input mode contains the following areas:

1   Input
2   Navigation and dialog keys
3   Copying and pasting
4   Switch to formula input

Formula input areas
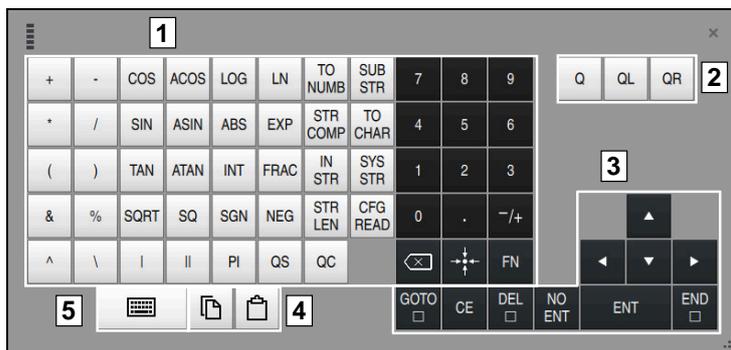
Virtual keyboard in formula input mode

Formula input mode contains the following areas:

1   Input
2   Q parameters
3   Navigation and dialog keys
4   Copying and pasting
5   Switch to NC input

11.1.1   Opening and closing the virtual keyboard

To open the virtual keyboard:

▶   Select the **virtual keyboard** on the control bar
>   The control opens the virtual keyboard.

To close the virtual keyboard:

▶   Select the **virtual keyboard** when the virtual keyboard is open

✕   ▶   Or press **Close** in the virtual keyboard
>   The control closes the virtual keyboard.

11.2 Message menu on the information bar

Application

In the message menu of the information bar, the control shows pending errors and notes. When opened, the control displays detailed information about the messages.

Description of function

The control uses the following symbols to differentiate between the types of messages:

Symbol	Message type	Meaning
❓	Error Question type	The control displays a dialog with several options you can select from. You cannot clear this error message: you can only choose one of the possible responses. If necessary, the control continues the dialog until the cause or correction of the error has been clearly determined.
⏻	Error Reset type	The control must be restarted. This message cannot be cleared.
⬤	Error Emergency-stop type	The control performs an emergency stop. An error message can only be cleared after the cause has been eliminated.
❗	Error	To continue, you must clear this message. An error message can only be cleared after the cause has been eliminated.
⚠	Warning	You can continue without clearing the message. Most warnings can be cleared at any time; in some cases, the cause has to be eliminated first.
ⓘ	Information	You can continue without clearing the message. You can clear the information at any time.
⚠	Note	You can continue without clearing the message. The control displays the note until you press the next valid key.
⬢		No pending messages

The message menu is collapsed by default.

The control displays messages upon various events, for example:

- Logical errors in the NC program
- Impossible contour elements
- Improper touch-probe inserts
- Hardware updates

Content
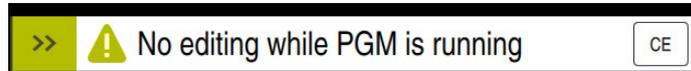
Collapsed message menu on the information bar

When the control displays a new message, the arrow to the left of the message blinks. Click or tap this arrow to confirm acknowledgment of the message; the control then minimizes the message.

The control displays the following information in the collapsed message menu:

- Message type
- Message
- Quantity of pending errors, warnings, and informational messages

Detailed messages

If you tap or click the symbol or within the message, the control expands the message menu.
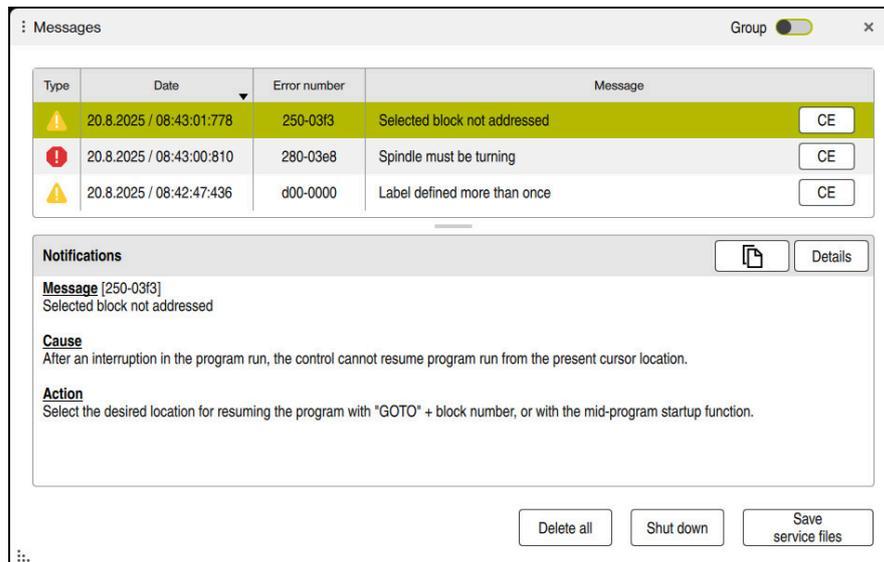
Expanded message menu with pending messages

The control displays all pending messages in chronological order.

The message menu shows the following information:

- Message type
- Date
- Error number
- Message
- Additional information (root cause, correction, information on the NC program)
  You can copy additional information to the clipboard with a button.

Deleting messages

Messages can be deleted in the following ways:

- **CE** key
- **CE** button in the message menu
- **Delete all** button in the message menu

Details

Press the **Details** button to show or hide internal information about the message. This information is of importance in case servicing is necessary.

Group

If you activate the **Group** toggle switch, the control displays all messages with the same error number in one row. This makes the list of messages shorter and easier to read.

Under the error number, the control displays the quantity of messages. Use **CE** to clear all messages of a group.

Service file

Click the **Save service files** button to open the **Save service files** window.

In the **Save service files** window, you can create service files in the following ways:

- If an error occurs, you can create a service file manually.

  Further information: "Creating a service file manually", Page 203

- If an error occurs repeatedly, a service file can be created automatically by means of the error number. Once the respective error occurs, the control saves a service file.

  Further information: "Creating a service file automatically", Page 204

Service files help service technicians in troubleshooting the problem. The control saves data that provide information about the current machine and operation status, such as active NC programs up to 10 MB, tool data, and keystroke logs.

The file name of each service file consists of a user-defined name and a timestamp.

If you create multiple service files with the same name, the control saves a maximum of five files and then deletes the file with the oldest timestamp, if necessary. Make a backup of the service files you created (e.g., by moving them to a different folder).

11.2.1 Creating a service file manually

To create a service file manually:
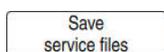
▲	▶ Expand the message menu
Save service files	▶ Select **Save service files**
	> The control opens the **Save service file** window.
	▶ Enter the file name
OK	▶ Select **OK**
	> The control saves the service file in the **TNC:\service** folder.

11.2.2   Creating a service file automatically

You can specify up to five error numbers for which the control will automatically create a service file if one of these errors occurs.

To specify a new error number:
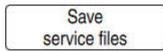
	▶ Expand the message menu
Save service files	▶ Select **Save service files**
	> The control opens the **Save service file** window.
Setting for autosave	▶ Select **Setting for autosave**
	> The control opens a table of error numbers.
	▶ Enter the desired error number
	▶ Select the **Active** checkbox
	> If the error occurs, the control automatically creates a service file.
	▶ Enter a comment, if applicable (e.g., to describe the problem)

12

The MDI application

Application

The **MDI** application allows you to execute individual NC blocks outside of the context of an NC program (e.g., for pre-positioning). When you press the **NC Start** key, the control will run the NC blocks separately.

You can also create an NC program step by step. The control remembers modally effective program information.

Related topics

- Creating NC programs

  Further information: Programming and Testing User's Manual

- Running NC programs

  Further information: "Program run", Page 229

Description of function

If you program using the millimeter unit of measurement, the control will use the NC program **$mdi.h** by default. If you program using the inch unit of measurement, the control will use the NC program **$mdi_inch.h**.
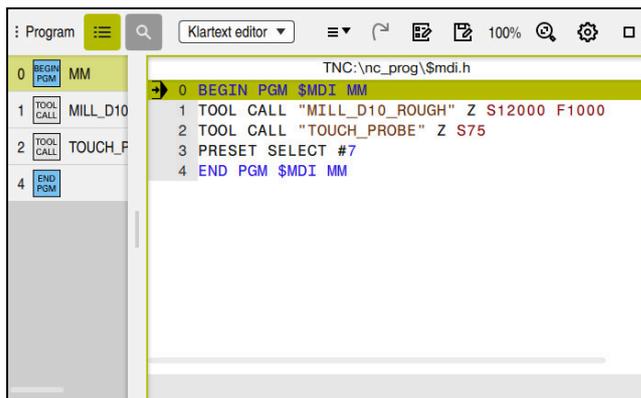
The **Program** workspace in the **MDI** application

The **MDI** application provides the following workspaces:

- **Document**

  Further information: Programming and Testing User's Manual

- **Help**

  Further information: Programming and Testing User's Manual

- **Positions**

  Further information: "The Positions workspace", Page 95

- **Program**

  Further information: Programming and Testing User's Manual

- **Simulation**

  Further information: Programming and Testing User's Manual

- **Status**

  Further information: "The Status workspace", Page 101

- **Keyboard**

  Further information: "Virtual keyboard of the control bar", Page 198

Icons and buttons

In the **MDI** application, the function bar provides the following buttons:

Icon or button	Meaning
⇥	Execution cursor The execution cursor shows which NC block is currently being executed or is marked for execution.
Insert NC function	The control opens the **Insert NC function** window. **Further information:** Programming and Testing User's Manual
Q info	The control opens the **Q parameter list** window, where you can see and edit the current values and descriptions of the variables. **Further information:** Programming and Testing User's Manual
GOTO block number	Mark an NC block to be run without considering any previous NC blocks **Further information:** Programming and Testing User's Manual
/ Skip block Off/On	Hide NC blocks with the **/** character. NC blocks hidden with a **/** character will be ignored during program run as soon as the **Skip block** toggle switch is active. **Further information:** Programming and Testing User's Manual
Skip block	If the toggle switch is active, the control does not execute NC blocks dimmed with a **/** character. If the toggle switch is active, then the control dims the NC blocks to be skipped. **Further information:** Programming and Testing User's Manual
; Comment Off/On	Insert or remove a **;** character in front of an NC block. If an NC block begins with a **;** character, then the block is a comment. **Further information:** Programming and Testing User's Manual
F LIMIT	Use this function to activate a feed-rate limit and define its value. **Further information:** "Feed rate limit F LIMIT", Page 234
Tool Retract	If the NC program is stopped during a thread cycle, you can retract the tool. **Further information:** Programming and Testing User's Manual
Edit	The control opens the context menu. **Further information:** Programming and Testing User's Manual
Tools	The control opens the **Tool management** application in the **Tables** operating mode. **Further information:** "Tool management ", Page 163
Internal stop	For example, if an NC program is interrupted due to an error or a stop, the control activates this button. Use this button to abort program run. **Further information:** "Interrupting, stopping or canceling program run", Page 235
Reset program	If you select **Internal stop**, the control activates this button. The control resets any modally active program information as well as the program run-time.

Modally effective program information

In the **MDI** application, you always execute NC blocks in **Single Block** mode. After the control has executed an NC block, program run is considered to be interrupted.

Further information: "Interrupting, stopping or canceling program run", Page 235

The block numbers of all NC blocks that you have successively run are shown in green.

The control saves the following data in this state:

- The last tool that was called
- Current coordinate transformations (e.g., datum shift, mirroring)

Notes

- In the **MDI** application, you can create and execute NC programs step by step. Then you can use **Save as** to save the current contents with a different file name.
- The following functions are not available in the **MDI** application:
  - Calling of an NC program with **PGM CALL**
  - Test run in the **Simulation** workspace
  - **Manual traverse** and **Approach position** functions while program run is interrupted
  - **Block scan** function
- The execution cursor is always displayed in the foreground. The execution cursor may cover or hide other icons.

13

Touch probe functions in the Manual operating mode (#17 / #1-05-1)

13.1 Fundamentals

Application

The touch probe functions allow you to set presets on the workpiece and perform measurements on the workpiece.

Related topics

- Preset table

  Further information: "Preset table *.pr", Page 295

- Datum table

  Further information: Programming and Testing User's Manual

- Reference systems

  Further information: "Reference systems", Page 172

- Preassigned variables

  Further information: Programming and Testing User's Manual

Requirements

- Touch Probe Functions software option (#17 / #1-05-1)

  > **i** If you are using a HEIDENHAIN touch probe with EnDat interface,
  > then the Touch Probe Functions software option (#17 / #1-05-1) is
  > automatically enabled.

- Calibrated workpiece touch probe

  Further information: "Calibrating the workpiece touch probe", Page 220

Description of function

The control provides the following functions for setting up the machine in the **Manual** operating mode in the **Setup** application:

- Setting a preset
- Calibrate the workpiece touch probe
- Calibrate the tool touch probe
- **Measure the tool**
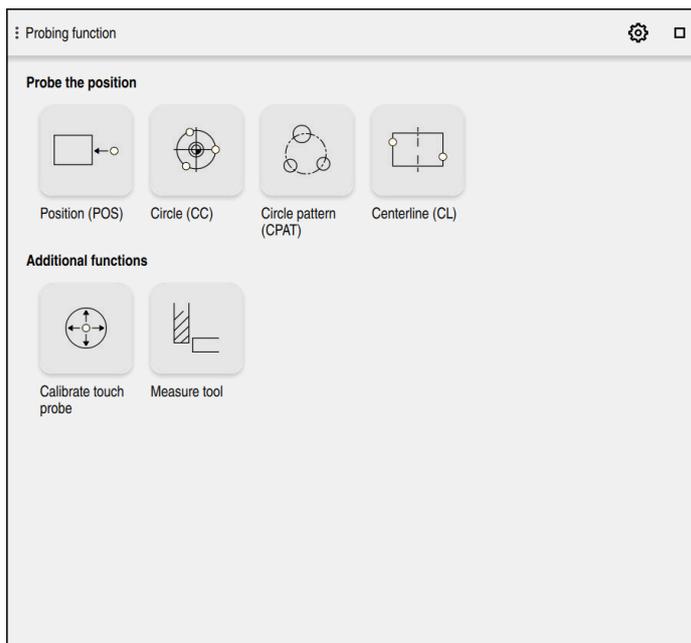
The **Probing function** workspace

Overview

The touch probe functions are structured in the following groups:

Probe the position

The **Probe the position** group contains the following touch probe functions:

Button	Function
Position (POS)	You can use the **Position (POS)** function to probe a position in the X axis, Y axis or Z axis.
	Further information: "Setting a preset in a linear axis", Page 217
Circle (CC)	The **Circle (CC)** function is used to determine the coordinates of a circle center point (e.g., for a hole or for a stud).
	Further information: "Setting the circle center point of a stud by means of the automatic probing method ", Page 218
Circle pattern (CPAT)	The **Circle pattern (CPAT)** function is used to determine the center point coordinates of a circle pattern.
Centerline (CL)	The **Centerline (CL)** function is used to determine the center point of a ridge or slot.

Additional functions

The **Additional functions** group contains the following touch probe functions:

Button	Function
Calibrate touch probe	The **Calibrate touch probe** function is used to determine the length and radius of a workpiece touch probe.
	Further information: "Calibrating the workpiece touch probe", Page 220
Measure tool	The **Measure tool** function allows you to measure tools by touch-off.
	Further information: "Measuring the tool by scratching", Page 225

Icons and buttons

General icons and buttons in the touch probe functions

The following icons and buttons are available, depending on the selected touch probe function:

Icon or button	Meaning
←	**Exit probing**
⊕	Select the workpiece preset and edit values, if required
	The control shows the number of the active preset to the right of the icon.
	Further information: "The Change the preset window", Page 215
	Further information: "Preset table *.pr", Page 295
ⓘ	Display help graphics for the selected touch probe function
Y+	Select the probing direction
⊹	**actual position capture**
○→▯	**Position**
	Manually approach and probe positions on a straight surface
	You position the workpiece touch probe and start the probing process manually.
⬡ A	**Circle center A**
	Automatically approach and probe positions on a stud or in a hole
	You manually pre-position the touch probe to the first probing point and define the parameters for the selected touch probe function. When you start the touch probe function, the control automatically positions and performs probing. After the defined number of touch points has been reached, the control determines the center of the stud or the hole.
	After the last probing process and if the opening angle contains the value 360°, the control positions the workpiece touch probe back to the position it had prior to starting the probing function.
⬡ M	**Circle center M**
	Automatically approach and probe positions on a stud or in a hole
	You position the workpiece touch probe and contact the individual probing points manually. If you select the **Assume measurement result** button, the control determines the center of the stud or hole.
Tools	The control opens the **Tool management** application in the **Tables** operating mode.
	Further information: "Tool management ", Page 163

Icon or button	Meaning
Handwheel	The control displays this toggle switch if a handwheel without display is configured on the control.
	If the handwheel is active, the operating mode's icon in the sidebar changes.
	Further information: "Electronic handwheel", Page 301
Internal stop	For example, if an NC program is interrupted due to an error or a stop, the control activates this button.
	Use this button to abort program run.
	Further information: "Interrupting, stopping or canceling program run", Page 235

Icons and buttons for calibration

The control offers the following functions for calibrating a 3D touch probe:

Icon or button	Meaning
⊥↓	Calibrating the length of a 3D touch probe
⊕ ⚬	Calibrating the radius of a 3D touch probe
Apply calibration data	Transferring values from the calibration process into tool management

Further information: "Calibrating the workpiece touch probe", Page 220

You can calibrate a 3D touch probe by using a calibration standard, such as a calibrating ring.

The control provides the following options:

Icon	Meaning
⨅	Measure the radius and the center offset using a calibration ring
⊓	Measure the radius and the center offset using a stud or a calibration pin
♀	Measure the radius and the center offset using a calibration sphere

Buttons for touch-probe functions

The control offers these buttons in the touch-probe functions during or after probing:

Button	Meaning
Remove last measurement	The control resets the last touch point. Only with the **Circle center M** measuring method
Assume measurement result	The control determines the circle center of the probed stud or the probed hole. The control enters the circle center in the **Measuring** column. Only with the **Circle center M** measuring method after at least three touch points.
Compensate the active preset	The control sets the active workpiece preset to the entered nominal value. If the workpiece preset is locked, the control shows a window with a confirmation request. **Further information:** "Preset table *.pr", Page 295
Correct the datum	The control opens the **Correct the datum** window. In the **Nominal value** area, enter the desired distance of the measuring result from the workpiece datum. Based on this information, the control calculates the required table value relative to the active workpiece preset. The control enters the calculated value in the selected line of the datum table. **Further information:** Programming and Testing User's Manual

The Change the preset window

In the **Change the preset** window you can select a preset or edit the values of a preset.

Further information: "Preset management", Page 179

The **Change the preset** window provides the following buttons:

Icon or button	Meaning
⌖	The control shows the preset table. **Further information:** Programming and Testing User's Manual
Apply changes and delete existing probe objects	The control activates the selected preset and rejects the touch points used so far. Then the control closes the window.
Apply	The control saves the changes and the selected preset. Then the control closes the window.
Reset	The control cancels the changes and restores the initial condition.
Cancel	The control closes the window without saving.

> **i** If you change a value, the control marks this value with a blue dot.
> If you accept the changes and the active preset is locked, the control shows a window with a confirmation request.

Log file of touch-probe functions

After executing any touch-probe function, the control writes the measured values to the TCHPRMAN.html file.

You can check the readings of past measurements in 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 directly under **TNC:**.

If you run several touch probe functions in a row, the control stores the measured values below each other.

13.1.1 Setting a preset in a linear axis

To probe the preset in any axis:

▶ Select the **Manual** operating mode

▶ Call the workpiece touch probe as a tool

▶ Select the **Setup** application

▶ Select **Position (POS)**

> The control opens the **Position (POS)** touch probe function.

▶ If necessary, check the values of the active preset or choose a different preset

▶ Use the axis keys to position the workpiece touch probe at the desired probing position (e.g., above the workpiece in the workspace)

▶ Select the probing direction (e.g., **Z-**)

▶ Press the **NC Start** key

> The control performs the probing process and then automatically retracts the workpiece touch probe to the starting point.

> The control shows the measurement results.

▶ In the **Nominal value** area, enter the new preset of the probed axis (e.g., **1**)

▶ Select **Compensate the active preset**

> The control sets the active preset to the entered nominal value.

> ℹ️ Using the **Position (POS)** touch probe function, you can probe in up to three axes.

▶ Select **Exit probing**

> The control closes the **Position (POS)** touch probe function.

13.1.2 Setting the circle center point of a stud by means of the automatic probing method

To probe a circle center point:

▶ Select the **Manual** operating mode

▶ Call the workpiece touch probe as a tool
▶ Select the **Setup** application

▶ Select **Circle (CC)**
> The control opens the **Circle (CC)** touch probe function.
▶ If necessary, check the values of the active preset or choose a different preset
▶ Position the workpiece touch probe at the desired probing position in the workspace
▶ Select **Type of contour** (e.g., stud)
▶ Enter **Diameter** (e.g., 60 mm)
▶ Enter the **Safety clearance (min. value = SET_UP)** if required

> ⓘ The control suggests the sum of the value in the **SET_UP** column of the touch probe table and the ball tip radius as a safety clearance.

▶ Position the workpiece touch probe at the desired probing position next to the workpiece and below the workpiece surface
▶ If necessary, turn the feed-rate potentiometer to zero
▶ Press the **NC Start** key

▶ If necessary, slowly turn up the feed-rate potentiometer
> The control executes the touch probe function based on the data entered.
> The control shows the measurement results.
> In the **Nominal value** area, enter the new preset of the probed axes (e.g., **0**)
▶ Select **Compensate the active preset**
> The control sets the active preset to the entered nominal value.
▶ Select **Exit probing**
> The control closes the **Circle (CC)** touch probe function.
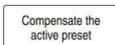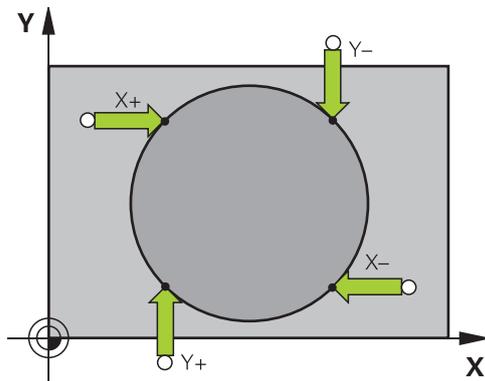
Notes

- If you use a non-contacting tool touch probe (such as a laser touch probe), then you are using touch-probe functions from a third-party supplier. Refer to your machine manual.
- In the touch probe functions, you can switch between the measuring methods **Position**, **Circle center A** and **Circle center M** as desired. With the **Intersection point (P)** function, you can switch the measuring method after two completed measurements.
- When probing while the guard door is open and spindle orientation to probing direction is active, the number of spindle revolutions is limited. When the maximum permitted number of spindle revolutions is reached, the direction of spindle rotation changes and the control may no longer orient the spindle on the shortest path.
- Use the **+**, **-**, *****, **/**, **(**, and **)** keys for calculations in the numerical input fields.
- 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 manufacturer has defined.
- When writing into an empty line of the preset table, the control automatically fills the other columns with values. To define a preset completely, you must determine the values in all axes and write them into the preset table.
- If no workpiece touch probe is inserted, the actual position can be captured with **NC Start**. The control displays a warning that no probing movement is carried out in that case.
- Recalibrate the workpiece touch probe in the cases below:
  - Initial configuration
  - Broken stylus
  - Stylus replacement
  - Change in the probe feed rate
  - Irregularities caused, for example, when the machine heats up
  - Changes to the **TRACK** parameter
- If the touch point is not reached during the probing process, the control will display a warning. The probing process can be continued with **NC Start**.

Definition

Spindle tracking

If the **Track** parameter in the touch probe table is active, the control orients the workpiece probing system so that the same position is always used for probing. By deflecting in the same direction, you can reduce the measurement error to the repeatability of the workpiece probing system. This behavior is called spindle tracking.

13.2 Calibrating the workpiece touch probe

Application

The touch probe must be calibrated in order to enable determining precisely the actual switching point of a 3D touch probe. Otherwise, the control cannot ascertain exact measuring results.

Related topics

- Touch probe table

  Further information: "Touch probe table tchprobe.tp (#17 / #1-05-1)", Page 285

Description of function
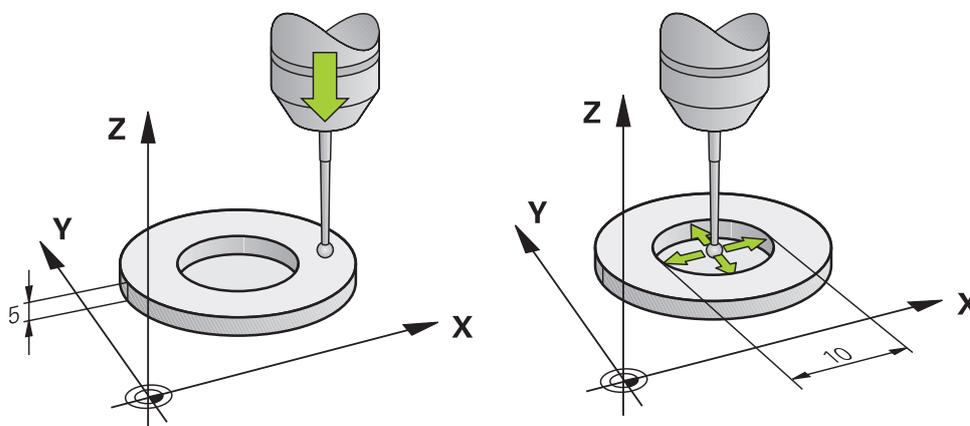
During calibration, the control finds the effective length of the stylus and the effective radius of the ball tip. To calibrate the 3D touch probe, clamp a calibration ring or a stud of known height and known radius to the machine table.

The effective length of the workpiece touch probe refers to the tool carrier preset.

Further information: "Tool carrier reference point", Page 151

You can calibrate the workpiece touch probe with various tools. For example, the workpiece touch probe can be calibrated using an overmilled surface in length and a calibration ring in the radius. This creates a reference between the workpiece touch probe and the tools in the spindle. In this procedure, measured tools and the calibrated workpiece touch probe correspond using the tool presetting device.

Calibrating an L-shaped stylus

Before you calibrate an L-shaped stylus you first must define the parameters in the touch probe table. Based on these approximate values, the control can align the touch probe during the calibration and determine the actual values.

First, define the following parameters in the touch probe table:

Parameter	Value to be defined
CAL_OF1	Length of extension
	The extension is the angled length of the L-shaped stylus.
CAL_OF2	0
CAL_ANG	Spindle angle at which the extension is parallel to the main axis
	For this, manually position the extension in the positive direction of the main axis and read the value from the position display.

After the calibration, the control overwrites the previously defined values in the touch probe table with the determined values.

Further information: "Touch probe table tchprobe.tp (#17 / #1-05-1)", Page 285

When calibrating the length, the control aligns the touch probe with the calibration angle defined in the **CAL_ANG** column.

While calibrating the touch probe, ensure that the feed rate override is 100%. That way you can always use the same feed rate for the subsequent probing processes as was used for the calibration. Hence, you can exclude inaccuracies during the probing caused by modified feed rates.

Reversal measurement

HEIDENHAIN touch probes are predefined as to whether or how a touch probe can be oriented. Other touch probes are configured by the machine manufacturer.

When calibrating the radius, up to three circular measurements can be taken depending on the possible orientation of the workpiece touch probe. The first two circular measurements determine the center offset of the workpiece touch probe. The third circular measurement determines the effective stylus tip radius. If orientation of the spindle is not possible or only a certain orientation is possible due to the workpiece touch probe, circular measurements are omitted.

13.2.1 Calibrating the length of the workpiece touch probe

To calibrate a workpiece touch probe using an overmilled surface in length:

▶ Measure the end milling cutter on the tool presetting device
▶ Store the measured end milling cutter in the tool magazine of the machine
▶ Enter the tool data of the end milling cutter in tool management
▶ Clamp the workpiece blank

▶ Select the **Manual** operating mode

▶ Replace the end milling cutter in the machine
▶ Switch on spindle (e.g., with **M3**)
▶ Use the handwheel to scratch the workpiece blank
   Further information: "Setting a preset with milling cutters", Page 179
▶ Set preset in the tool axis (e.g., with **Z**)
▶ Position the end milling cutter next to the workpiece blank
▶ Set a small value in the tool axis (e.g., **-0.5 mm**)
▶ Overmill the workpiece blank using the handwheel
▶ Set the preset again in the tool axis (e.g., with **Z=0**)
▶ Switch off spindle (e.g., with **M5**)
▶ Insert the workpiece touch probe

▶ Select the **Setup** application

▶ Select **Calibrate touch probe**

▶ If necessary, check the values of the active preset or choose a different preset

▶ Select measuring method **Length**:
> The control displays the current calibration data.
▶ Enter a **Position of reference surface** (e.g., **0**)
▶ Position the workpiece touch probe close to the surface of the overmilled area

> ⓘ Check that the area to be probed is flat and free of chips before you start the touch probe function.

▶ Press the **NC Start** key
> The control performs the probing operation and then automatically retracts the workpiece touch probe to the starting point.
▶ Check results

Apply calibration data
▶ Select **Apply calibration data**
> The control transfers the calibrated length of the 3D touch probe to the tool table.

▶ Select **Exit probing**
> The control closes the **Calibrate touch probe** touch probe function.

13.2.2 Calibrating the radius of the workpiece touch probe

To calibrate a workpiece touch probe using a calibration ring in the radius:

▶ Clamp the calibration ring on the machine table (e.g., with clamps)

▶ Select the **Manual** operating mode

▶ Position the workpiece touch probe in the hole of the calibration ring

> ⓘ Make sure that the stylus tip is completely recessed into the calibration ring. This causes the control to probe using the largest circumference of the calibration sphere.

▶ Select the **Setup** application

▶ Select **Calibrate touch probe**

▶ If necessary, check the values of the active preset or choose a different preset

▶ Select measuring method **Radius**:

▶ Select calibration standard **Calibration ring**

▶ Enter the **Diameter** of the calibration ring

▶ Enter the **Safety clearance (min. value = SET_UP)** if required

▶ Press the **NC Start** key

> The workpiece touch probe probes all required touch points in an automatic probing routine. The control calculates the effective stylus tip radius. If probing from opposite orientations is possible, the control calculates the center offset.

▶ Check results

Apply calibration data
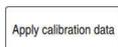
▶ Select **Apply calibration data**

> The control stores the calibrated radius of the touch probe in the tool table.

▶ Select **Exit probing**

> The control closes the **Calibrate touch probe** touch probe function.

Instructions for calibration

■ In order to be able to determine ball-tip center misalignment, the control needs to be specially prepared by the machine manufacturer.

■ If you press the **OK** button after the calibration process, the control accepts the calibration values for the active touch probe. The updated tool data then becomes immediately effective, and it is not necessary to repeat the tool call.

■ HEIDENHAIN guarantees the proper operation of the touch probe cycles only in conjunction with HEIDENHAIN touch probes.

■ If you want to calibrate using the outside of an object, you need to pre-position the touch probe above the center of the calibration sphere or calibration pin. Ensure that the probing points can be approached without collisions.

■ Use the **+**, **-**, *****, **/**, **(**, and **)** keys for calculations in the numerical input fields.

■ 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 in the touch probe table. The control uses the **TP_NO** parameter to link the data from the touch probe table with the data from the tool table.

Further information: "Touch probe table tchprobe.tp (#17 / #1-05-1)", Page 285

13.3 Measuring the tool by scratching

Application

Not all machines are equipped with a tool touch probe for measuring a tool. The **Tool measured** touch probe function enables determining the tool dimensions by scratching a workpiece.

Related topics

- Touch probe functions in the **Setup** application

  Further information: "Touch probe functions in the Manual operating mode (#17 / #1-05-1)", Page 209

- Measuring the tool automatically with cycles

  Further information: Programming and Testing User's Manual

Requirement

- Touch Probe Functions software option (#17 / #1-05-1)

  > **ⓘ** If you are using a HEIDENHAIN touch probe with EnDat interface, then the Touch Probe Functions software option (#17 / #1-05-1) is automatically enabled.

Description of function

You do not use a 3D touch probe for scratching, but the tool to be measured. In the scratching process, approach the tool carefully to a workpiece surface until you can see a thin chip being removed. The handwheel allows obtaining a higher accuracy.

In the **X** or **Y** probing directions, the tool radius can be determined. When selecting probing direction **Z**, the tool length is measured.

Buttons in the Measure the tool function

The control offers the following options for writing the measured radius or length values into the tool table:

Button	Meaning
Write basic values	The control transfers the values into columns **R** or **L**. The control resets existing delta values in columns **DR** or **DL**.
Write delta values	The control enters the delta values in columns **DR** or **DL**.

Further information: "Tool tables", Page 278

13.3.1 Tool measurement by scratching

The dimensions of an end mill can be determined by using the **Tool measured** function as follows:

▶ Select the **Manual** operating mode

▶ Set the workpiece preset if required

> 🛈 Position the workpiece preset on the surfaces to be scratched in order to obtain a clear reference.

▶ Insert the tool to be measured
▶ Define the speed if required
▶ Start the tool spindle
▶ Select the **Setup** application

▶ Select the **Measure tool** touch probe function

▶ If necessary, check the values of the active preset or choose a different preset
▶ Scratch the workpiece in the desired axis direction (e.g., **X+**)

▶ Select the associated probing direction **X+**

▶ Select **actual position capture**
> The control transfers the actual X axis position into the **Actual value** column.
> The control shows the measurement results.
▶ Enter a **Nominal value** (e.g., **0**)
▶ Select **Write basic values**
> The control transfers the value into column **R** of the tool table.
> The control resets the existing delta value in the **DR** column.

> 🛈 When selecting **Write delta values**, the control will enter only one delta value in column **DR**.

▶ Scratch another axis direction if required (e.g., **Z-**)

▶ Select **Exit probing**
> The control closes the **Measure tool** touch probe function.

13.4   Suppressing touch probe monitoring

Application

If you move a workpiece touch probe too close to the workpiece, you can accidentally deflect the workpiece touch probe. You cannot retract a deflected workpiece touch probe in the monitored state. You can retract a deflected workpiece touch probe by suppressing touch probe monitoring.

Description of function

If the control does not receive a stable signal from the probe, the button displays **Suppress touch probe monitoring**.

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 remains active only for 30 seconds.

13.4.1   Deactivating touch probe monitoring

To deactivate touch probe monitoring:

▶ Select the **Manual** operating mode

▶ Select **Suppress touch probe monitoring**
> 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.

Notes

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** operating mode

If the touch probe sends a stable signal within these 30 seconds, then touch-probe monitoring reactivates itself automatically and the error message is cleared.

14

Program run

14.1 The Program Run operating mode

14.1.1 Fundamentals

Application

In the **Program Run** operating mode you produce workpieces by having the control execute NC programs either block-by-block or in full sequence.

Related topics

- Executing individual NC blocks in the **MDI** application
  Further information: "The MDI application ", Page 205
- Creating NC programs
  Further information: Programming and Testing User's Manual

NOTICE

Caution: Danger due to manipulated data!

If you execute NC programs directly from a network drive or a USB device, you have no control over whether the NC program has been changed or manipulated. In addition, the network speed can slow down the execution of the NC program. Undesirable machine movements or collisions may result.

▶ Copy the NC program and all called files to the **TNC:** drive

---

NOTICE

Danger of collision!

When you edit NC programs outside the **Program** workspace, you have no control over whether the control will identify the changes. Undesirable machine movements or collisions may result.

▶ Edit NC programs in the **Program** workspace only

Description of function

When you select a new NC program or when an NC program has been completely executed, the cursor is at the beginning of the program.

If you want to start machining at a different NC block, you first need to select the desired NC block by using the **Block scan** function.

Further information: "Block scan for mid-program startup", Page 240

By default, the control runs NC programs in Full Sequence mode after the **NC Start** key has been pressed. In this mode, the control runs an NC program continuously up to its end, or up to a manual or programmed interruption.

In **Single Block** mode you execute each NC block separately by pressing the **NC Start** key.

The control shows the status of the machining process with the **Control-in-operation** icon in the status overview.

Further information: "Status overview of the TNC bar", Page 99

The **Program Run** operating mode provides the following workspaces:

■ **Document**

Further information: Programming and Testing User's Manual

■ **Positions**

Further information: "The Positions workspace", Page 95

■ **Program**

Further information: Programming and Testing User's Manual

■ **Simulation**

Further information: Programming and Testing User's Manual

■ **Status**

Further information: "The Status workspace", Page 101

Icons and buttons

The **Program Run** operating mode contains the following icons and buttons:

Icon or button	Meaning
⬜	**Open File** With **Open File** you can open a file (for example, an NC program). When you open a file, the control closes the file that was already open.
⮕	Execution cursor The execution cursor shows which NC block is currently being executed or is marked for execution.
Single Block	If this toggle switch is active, then you run each NC block separately with the **NC Start** key. If Single Block mode is selected, then the operating mode's icon in the control bar changes.
Q info	The control opens the **Q parameter list** window, where you can see and edit the current values and descriptions of the variables. **Further information:** Programming and Testing User's Manual
Compensation tables	The control opens a selection menu with the following tables: ■ **D** ■ **T-CS** ■ **WPL-CS** **Further information:** "Compensation during program run", Page 251
F LIMIT	Use this function to activate a feed-rate limit and define its value. **Further information:** "Feed rate limit F LIMIT", Page 234
Program run options	When you select this button, the control opens the **Program run options** window with the following selection possibilities: ■ **Feed F LIMIT** Use this function to activate a feed-rate limit and define its value. **Further information:** "Feed rate limit F LIMIT", Page 234 ■ **Skip block** If the toggle switch is active, the control does not execute NC blocks dimmed with a **/** character. If the toggle switch is active, then the control dims the NC blocks to be skipped. **Further information:** Programming and Testing User's Manual ■ **Pause at M1** If the toggle switch is active, the control stops the execution at every NC block with **M1**. If the toggle switch is inactive, then the control dims the **M1** syntax element. **Further information:** Programming and Testing User's Manual
Skip block	If the toggle switch is active, the control does not execute NC blocks dimmed with a **/** character. If the toggle switch is active, then the control dims the NC blocks to be skipped. **Further information:** Programming and Testing User's Manual

Icon or button	Meaning
Pause at M1	If the toggle switch is active, the control stops the execution at every NC block with **M1**.
	If the toggle switch is inactive, then the control dims the **M1** syntax element.
	Further information: Programming and Testing User's Manual
Automatic program start	Starts machining at a defined time automatically
	Further information: "Automatic program start", Page 248
GOTO block number	Mark an NC block to be run without considering any previous NC blocks
	Further information: Programming and Testing User's Manual
Manual traverse	While a program run is interrupted, you can move the axes manually.
	If **Manual traverse** is active, the operating mode's icon in the control bar changes.
	Further information: "Manual traverse during an interruption", Page 239
Handwheel	If **Manual traverse** is active, you can activate and deactivate the handwheel with this switch.
	The keys of the machine operating panel cannot be used for traversing while the handwheel is active.
	Further information: "Manual traverse during an interruption", Page 239
Approach position	Return to contour after manual traverse of the machine axes during an interruption
	Further information: "Returning to the contour", Page 247
Block scan	Use the **Block scan** function to start program run at any desired NC block.
	The control takes the preceding parts of the NC program up to this NC block into account mathematically; for example, whether the spindle was switched on with **M3**.
	Further information: "Block scan for mid-program startup", Page 240
Tool Retract	If the NC program is stopped during a thread cycle, you can retract the tool.
	Further information: "The Retract application", Page 252
Open in the editor	The control opens the active NC program in the **Editor** operating mode and selects the currently selected NC block, even for called NC programs.
	Further information: Programming and Testing User's Manual
Tools	The control opens the **Tool management** application in the **Tables** operating mode.
	Further information: "Tool management ", Page 163
Internal stop	For example, if an NC program is interrupted due to an error or a stop, the control activates this button.
	Use this button to abort program run.
Reset program	If you select **Internal stop**, the control activates this button.
	The control places the cursor back to the beginning of the program and resets any modally active program information as well as the program run-time.

Feed rate limit F LIMIT

The **F LIMIT** button allows you to reduce the feed rate for all operating modes. The reduction applies to all rapid traverse and feed rate movements. The value you have entered remains active across power cycles.

The **F LIMIT** button is available in the **MDI** application and in the **Editor** operating mode.

When you select the **F LIMIT** button in the function bar, the control will open the **Feed rate F LIMIT** window.

Use the **+**, **-**, *****, **/**, **(**, and **)** keys for calculations in the numerical input fields.

If a feed rate limit is active, the control highlights the **F LIMIT** button in color and displays the defined value. In the **Positions** and **Status** workspaces, the feed rate is displayed in orange.

Further information: "Status displays", Page 93

You deactivate the feed rate limit by entering a value of 0 in the **Feed rate F LIMIT** window.

Interrupting, stopping or canceling program run

There are several ways to stop a program run:

- Interrupt program run (e.g., with the miscellaneous function **M1**)
- Stop program run (e.g., with the **NC Stop** key)
- Cancel program run (e.g., with the **NC stop** key and the **Internal stop** button)
- Terminate program run (e.g., with the miscellaneous functions **M2** or **M30**)

Upon major errors, the control automatically aborts program run (e.g., during a cycle call with stationary spindle).

Further information: "Message menu on the information bar", Page 201

If you run your NC program in **Single Block** mode or in the **MDI** application, the control will switch to the interrupted state after the execution of each NC block.

The control shows the current program run status with the **Control-in-operation** icon.

Further information: "Status overview of the TNC bar", Page 99

Below are some of the functions you can execute in an interrupted state:

- Select the **Manual** operating mode
- The **Manual traverse** function
- Checking variables and changing these if necessary using the **Q INFO** function
- Changing the setting for the optional programmed interruption with **M1**
- Changing the setting for the programmed skipping of NC blocks with **/**

> **i** You can use the **Manual traverse** function also when program run is stopped.

NOTICE

Danger of collision!

Certain manual interactions may lead to the control losing the modally effective program information (i.e., the contextual reference). Loss of this contextual reference may result in unexpected and undesirable movements. There is a risk of collision during the subsequent machining operation!

- ▶ 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 the values of variables by using the **Q parameter list** window
  - Switching the operating modes
- ▶ Restore the contextual reference by repeating the required NC blocks

---

Programmed interruptions

You can set interruptions directly in the NC program. The control interrupts program run in the NC block containing one of the following inputs:

- Programmed stop **STOP**
- Conditional stop **M1**

Resuming program run

After stopping the program with the **NC Stop** key or a programmed interruption, you can resume program run by pressing the **NC Start** key.

After canceling program run with an **Internal stop**, you must start program run at the beginning of the NC program or use the **Block scan** function.

After an interruption of program run within a subprogram or program section repeat, you need to use the **Block scan** function for mid-program startup.

Further information: "Block scan for mid-program startup", Page 240

Modally effective program information

The control saves the following data during a program interruption:

■ The last tool that was called
■ Current coordinate transformations (e.g., datum shift, mirroring)
■ The coordinates of the circle center that was last defined

The control uses the stored data for returning the tool to the contour (**Approach position** button).

Further information: "Returning to the contour", Page 247

> **i** The saved data remains active until it is reset (e.g., by selecting a program).

Notes

> **NOTICE**
>
> **Danger of collision!**
>
> Program cancellation, manual intervention, forgotten resetting of NC functions or transformations can lead to the control performing unexpected or undesirable movements. This can lead to workpiece damage or collision.
>
> ▶ Rescind all programmed NC functions and transformations within the NC program
> ▶ Run a simulation before executing an NC program
> ▶ Check both the general as well as the additional status display for NC functions and transformations, such as an active basic rotation, before executing an NC program
> ▶ Carefully prove-out the NC programs in **Single Block** mode

- In the **Program Run** operating mode, the control marks active files with the status **M**, such as a selected NC program or tables. If you open such a file in another operating mode, the controls shows the status on the tab of the application bar.
- When positioning an axis, the control checks whether the defined speed has been reached. The control does not check the speed in positioning blocks where **FMAX** is the feed rate.
- You can adjust the feed rate and the spindle speed during program run with the potentiometers.
- If you modify the workpiece preset during a program run interruption, you must re-select the NC block to resume.
  Further information: "Block scan for mid-program startup", Page 240
- HEIDENHAIN recommends switching the spindle on with **M3** or **M4** after every tool call. That way you avoid problems during program run, such as when restarting after an interruption.
- The execution cursor is always displayed in the foreground. The execution cursor may cover or hide other icons.

14.1.2 Navigation path in the Program workspace

Application

If you execute an NC program, or if you test it in the opened **Simulation** workspace, the control will display a navigation path in the file information bar of the **Program** workspace.

The control displays the names of all the NC programs used in the navigation path and opens the contents of all NC programs in the workspace. This makes it easier to keep an overview of the execution when calling programs and allows navigating between the NC programs when program run is interrupted.

Related topics

■ Program call

Further information: Programming and Testing User's Manual

■ The **Program** workspace

Further information: Programming and Testing User's Manual

■ The **Simulation** workspace

Further information: Programming and Testing User's Manual

■ Interrupted program run

Further information: "Interrupting, stopping or canceling program run", Page 235

Requirement

■ The **Program** and **Simulation** workspaces are both opened

In the **Editor** operating mode you need both workspaces to use the function.

Description of function

The control shows the name of the NC program as a path element in the file information bar. As soon as the control calls a different NC program, the control adds a new path element with the name of the called NC program to the bar.

Additionally, the control displays the contents of the called NC program in a new pane in the **Program** workspace. The control displays as many NC programs side by side as the size of the workspace permits. If necessary, newly opened NC programs will cover previously opened NC programs. The control displays the covered NC programs in a narrow band at the left edge of the workspace.

When execution is interrupted, you can navigate between the NC programs. When you select the path element of an NC program, the control opens the content.

When you select the last path element, the control automatically marks the active NC block with the execution cursor. When you press the **NC Start** key, the control resumes execution of the NC program from this position.
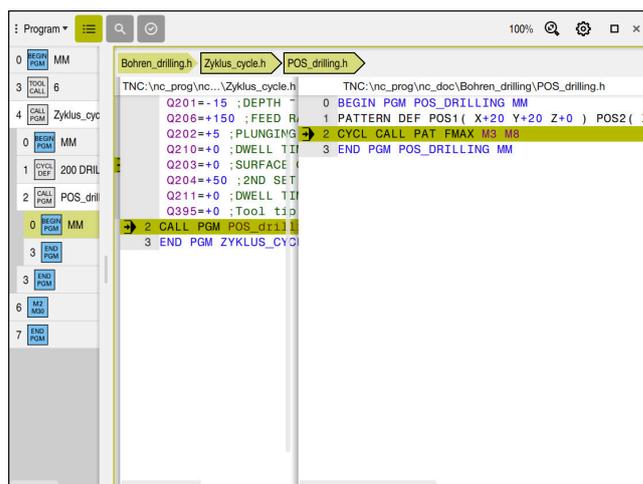
Called NC programs in the **Program** workspace in the **Program Run** operating mode

Depiction of path elements

The control displays the path elements of the navigation path as follows:

Format	Meaning
Black frame	The NC program is visible in the **Program** workspace and is not covered by other NC programs.
Highlighted in green	The NC program at the current cursor position is active or is considered for program run. If, for example, the cursor is positioned in the called NC program, the calling NC program will be considered for program run.
Highlighted in gray	The NC program is active for execution but will not be considered for program run at the current cursor position. If, for example, you stop the execution and navigate into the calling NC program, the control displays the path element of the called NC program in gray.

Note

In the **Program Run** operating mode, the **Structure** column contains all structuring items, even those of the called NC programs. The control indents the structure of the called NC programs.

The structure items allow you to navigate into every NC program. The control displays the associated NC programs in the **Program** workspace. The navigation path always remains at the current point of execution.

Further information: Programming and Testing User's Manual

14.1.3 Manual traverse during an interruption

Application

You can interrupt program run and move the machine axes manually, for example to reduce the risk of injury during workpiece measurement.

Related topics

- Manual traverse of machine axes

  Further information: "Moving the machine axes", Page 126

Description of function

When you select the **Manual traverse** button, you can move the axes of the machine.

Further information: "Using axis keys to move the axes ", Page 128

If **Manual traverse** is active, the operating mode's icon in the control bar changes.

Notes

- Refer to your machine manual.

  On some machines, you may have to press the **NC Start** key while **Manual traverse** is active in order to enable the axis keys.

- Using the optional machine parameter **userControl** (no. 144101), the machine manufacturer defines if the control will display the **Handwheel** toggle switch or not.

  If the **Handwheel** toggle switch is available, the control displays an additional area in its machine settings. You can define whether the machine operating panel or the handwheel is active by default.

  Further information: "The Machine Settings menu item", Page 333

  If this machine parameter has been defined with **FALSE** or not at all, the machine manufacturer determines the user control that is active by default.

- While the **Handwheel** toggle switch is active, you cannot move the axes with the machine operating panel.

- Refer to your machine manual.

  Depending on the machine, the control indicates that a handwheel is active (for example, with a handwheel symbol next to the selected axis in the **Positions** workspace or via a message in the information bar).

14.1.4 Block scan for mid-program startup

Application

The **Block scan** function allows you to start an NC program at any desired NC block. The control factors workpiece machining up to this NC block into the calculations. For example, the control will switch on the spindle before the start.

Related topics

- Creating NC programs

  Further information: Programming and Testing User's Manual

Requirement

- The function must be enabled by your machine manufacturer.

  The **Block scan** function must be enabled and configured by your machine manufacturer.

Description of function

If the NC program was interrupted under the following conditions, the control saves the interruption point:

- The **Internal stop** button
- Emergency stop
- Power failure

If, while restarting, the control finds a saved point of interruption, then it outputs a message. You can then execute a block scan directly to the point of interruption. The control displays the message when you switch to **Program Run** operating mode for the first time.

You have the following options for a block scan:

- Block scan in the main program, with repetitions if necessary

  Further information: "Performing a single-level block scan", Page 243
- Multi-level block scan in subprograms and touch probe cycles

  Further information: "Performing a multi-level block scan", Page 244
- Block scan in a point table

  Further information: "Block scan in point tables", Page 245

At the start of the block scan, the control resets the data, as with a selection of a new NC program. During the block scan you can activate or deactivate **Single Block** mode.

The Block scan window

The **Block scan** window with saved interruption point and open **Advanced** area

The **Block scan** window provides the following data:

Row	Meaning
Program	Path of the active NC program
	Path of the NC program that is active at the time of interruption
Block number	Number of the NC block at which program run should start
	The control proposes the currently selected NC block. You can open the **Search** column from the **Block scan** window.
	Number of the NC block that was active at the time of interruption
Repetitions	Number of the repetition for mid-program startup if the desired NC block is located within a program-section repeat or a program loop.
	Number of the repetition at the time of interruption if the interruption was located within a program-section repeat or program loop.
Point table	Path of the point table Selection by means of a selection window
	Path of the active point table at the time of interruption
Point number	Row in the point table
	Active row in the point table at the time of interruption

Select the interruption point by using the **Select last** button in the **Advanced** area.

Performing a single-level block scan

To start in an NC program by using a single-level block scan:

▶ Select the **Program Run** operating mode

Block scan
▶ Select **Block scan**
> The control opens the **Block scan** window. The fields **Program**, **Block number** and **Repetitions** contain the current values.
▶ Enter the **Program** if required
▶ Enter **Block number**
▶ Enter the **Repetitions** if required
▶ If applicable, open the **Extended** area
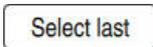
Select last
▶ If required, use **Select last** to start at a saved interruption point

▶ Press the **NC Start** key
> The control starts the block scan and calculates until the entered NC block.
> If you have changed the machine status, the control displays the **Restore machine status** window.

▶ Press the **NC Start** key
> The control restores the machine status (e.g., **TOOL CALL** or M functions).
> If you have changed the axis positions, the control displays the **Axis sequence for return to contour:** window.

▶ Press the **NC Start** key
> Using the displayed positioning logic, the control moves to the required positions.

> **ⓘ** You can also position the axes individually in your own selected sequence.
> **Further information:** "Positioning the axes in a self-selected sequence", Page 248

▶ Press the **NC Start** key
> The control resumes execution of the NC program.

Performing a multi-level block scan

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, you first go to the desired subprogram call and then continue the block scan. The same procedure is used for called NC programs.

To start in an NC program by using a multi-level block scan:

▶ Select the **Program Run** operating mode

▶ Select **Block scan**

> The control opens the **Block scan** window. The fields **Program**, **Block number** and **Repetitions** contain the current values.

▶ Perform a block scan to the first start-up point

Further information: "Performing a single-level block scan", Page 243

▶ Activate the **Single Block** toggle switch as needed

▶ Press the **NC Start** key to execute individual NC blocks as needed

▶ Select **Continue block scan**

▶ Define the NC block where you wish to start

▶ Press the **NC Start** key

> The control starts the block scan and calculates until the entered NC block.

> If you have changed the machine status, the control displays the **Restore machine status** window.

▶ Press the **NC Start** key

> The control restores the machine status (e.g., **TOOL CALL** or M functions).

> If you have changed the axis positions, the control displays the **Axis sequence for return to contour:** window.

▶ Press the **NC Start** key

> Using the displayed positioning logic, the control moves to the required positions.

> ℹ️ You can also position the axes individually in your own selected sequence.
> **Further information:** "Positioning the axes in a self-selected sequence", Page 248

▶ Select **Continue block scan** again as needed

▶ Repeat the steps

▶ Press the **NC Start** key

> The control resumes execution of the NC program.

Block scan in point tables

To start in a point table:

▶ Select the **Program Run** operating mode

Block scan
▶ Select **Block scan**
> The control opens the **Block scan** window. The fields **Program**, **Block number** and **Repetitions** contain the current values.

▶ Open the **Extended** area

▶ **Point table**: Select the point table

▶ **Point number**: Select the row number of the point table for mid-program startup

▶ Press the **NC Start** key
> The control starts the block scan and calculates until the entered NC block.
> If you have changed the machine status, the control displays the **Restore machine status** window.

▶ Press the **NC Start** key
> The control restores the machine status (e.g., **TOOL CALL** or M functions).
> If you have changed the axis positions, the control displays the **Axis sequence for return to contour:** window.

▶ Press the **NC Start** key
> Using the displayed positioning logic, the control moves to the required positions.

> **i** You can also position the axes individually in your own selected sequence.
> **Further information:** "Positioning the axes in a self-selected sequence", Page 248

> **i** If you would like to use the block scan function to start in a point pattern, then use the same procedure. Define the desired start-up point in the **Point number** field. The first point in the point pattern has the number 0.
> **Further information:** Programming and Testing User's Manual

Notes

> ## *NOTICE*
>
> **Danger of collision!**
>
> If you select an NC block in program run using the **GOTO** function and then execute the NC program, the control ignores all previously programmed NC functions (e.g., transformations). This means that there is a risk of collision during subsequent traversing movements!
>
> ▶ Use **GOTO** only when programming and testing NC programs
> ▶ Only use **Block scan** when executing NC programs

> ## *NOTICE*
>
> **Caution: Danger to the tool and workpiece!**
>
> If program run is interrupted within a called NC program, the control always offers the first call of this NC program as a point of interruption for mid-program startup. If program run was interrupted in a later call, the control might then execute parts of the program that have already run.
>
> ▶ Use multi-level block scan to manually navigate to the point of interruption

> ## *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

- The TNC7 go can move up to four axes simultaneously. If an NC block commands movement of more than four axes, the control displays an error message. When the control reads such an NC block during the **Block scan**, it will also display an error message.
- The control only displays the dialogs required by the process in the pop-up window.
- Use the **+**, **-**, *****, **/**, **(**, and **)** keys for calculations in the numerical input fields.
- Even after an internal stop, the control shows the number of repetitions on the **LBL** tab of the **Status** workspace.

  Further information: "The LBL tab", Page 103
- HEIDENHAIN recommends switching the spindle on with **M3** or **M4** after every tool call. That way you avoid problems during program run, such as when restarting after an interruption.

14.1.5 Returning to the contour

Application

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 after a block scan (e.g., 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

Related topics

- Manual traverse during program run interruptions
  Further information: "Manual traverse during an interruption", Page 239
- The **Block scan** function
  Further information: "Block scan for mid-program startup", Page 240

Description of function

If you have selected the **Manual traverse** button, this button will change to **Approach position**.

When you select **Approach position**, the control will open the **Axis sequence for return to contour:** window.

The **Axis sequence for return to contour: window**
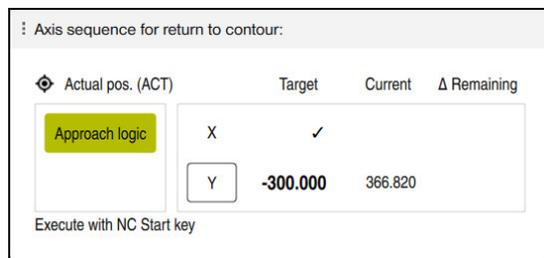
The **Axis sequence for return to contour:** window

In the **Axis sequence for return to contour:** window, the control displays all of the axes that are not yet located at the correct position for program execution.

The control suggests a positioning logic for the sequence of the traversing movements. If the tool is located in the tool axis below the position to be approached, then the control offers the tool axis as the first traverse direction. You can also traverse the axes in a self-selected sequence.

Further information: "Positioning the axes in a self-selected sequence", Page 248

If manual axes are included in the axes to be returned to the contour, then the control will not suggest a positioning logic. As soon as you have correctly positioned the manual axis, the control will suggest a positioning logic for the remaining axes.

Further information: "Positioning manual axes", Page 248

Positioning the axes in a self-selected sequence

To position the axes in a self-selected sequence:

> Approach
> position

▶ Select **Approach position**
> The control displays the **Axis sequence for return to contour:** window and the axes to be positioned.
▶ Select the desired axis (e.g., **X**)

▶ Press the **NC Start** key
> The control moves the axis to the required position.
> When the axis has reached the correct position, the control shows a check mark for **Target**.
▶ Position the remaining axes
> When all axes have reached their positions, the control closes the window.

Positioning manual axes

To position manual axes:

> Approach
> position

▶ Select **Approach position**
> The control displays the **Axis sequence for return to contour:** window and the axes to be positioned.
▶ Select the manual axis (e.g., **W**)
▶ Position the manual axis to the value shown in the window
> When a manual axis with encoder has reached the position, the control automatically clears the value.
▶ Select **Axis in position**
> The control saves the position.

Note

In the machine parameter **restoreAxis** (no. 200305), the machine manufacturer defines in which sequence of axes the control approaches the contour again.

Definition

Manual axis
Manual axes are non-driven axes that need to be positioned by the machine operator.

14.1.6 Automatic program start

Application

With the **Automatic program start** function you define a specific time at which the control will start to execute an NC program, such as a warm-up program for the machine. No machine operator needs to be present.

Requirements

- The function must be enabled by your machine manufacturer.

  In the machine parameter **autoStartEnabled** (no. 100701), the machine manufacturer defines whether an automatic program start is possible.

- Machine is switched on

- No NC program is currently running on the machine

  If a program is already running at the specified time, the control will not execute an automatic program start.

- Machine is prepared for the NC program (e.g., correct workpiece preset is active)

  > **i** HEIDENHAIN recommends activating the workpiece preset in the NC program.
  >
  > **Further information:** Programming and Testing User's Manual

Description of function

Use this function to execute NC programs.

When you select the **Automatic program start** button, the control displays the **Automatic program start** window.
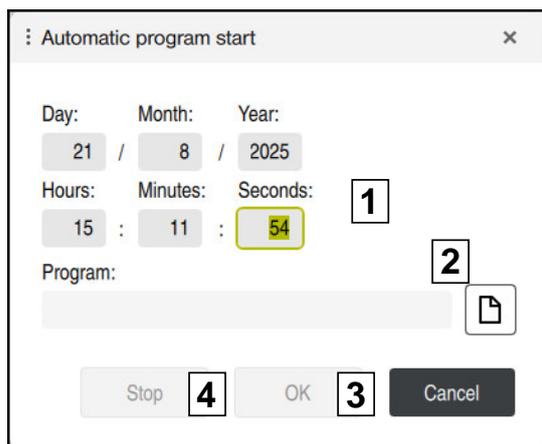
The **Automatic program start** window

The control offers the following possibilities in the **Automatic program start** window:

1 Defining the date and time
2 Selecting an NC program via a selection window
3 Activating automatic program start
4 Deactivating automatic program start

If an automatic program start is active, the control displays an icon in the information bar.

Further information: "Icons on the control's user interface", Page 73

Notes

⚠ WARNING

Caution: hazard to the user!

If automatic program start is active, the machine starts an NC program on its own and moves the axes. There is an increased risk of injury on machines without housing.

▶ Clearly indicate on the machine whether **Automatic program start** is active
▶ If necessary, restrict access to the machine

NOTICE

Caution: Significant property damage!

If automatic program start is active, the machine starts an NC program without a machinist and moves the axes. Collisions can occur if the machine was not set up correctly, or if there have been changes in the meantime. There might be no machinist present in order to abort program run.

▶ Set up the machine appropriately for such machining, e.g.:
  ▪ Install the correct workholding equipment
  ▪ Measure the tools used in the NC program and stock them in the magazine
▶ Optimize the NC program for automation, e.g.:
  ▪ Automatic presetting
  ▪ Activating the most recently set preset
  ▪ Calling the correct workholding equipment

▪ In the machine parameter **closeDialogOnOK** (no. 100702), the machine manufacturer defines whether the control closes the **Automatic program start** window after activation.
▪ In the machine parameter **useLastStartData** (no. 100703), the machine manufacturer defines whether the control offers the last defined settings when opening the **Automatic program start** window.
▪ Ensure that the tools being used have sufficient service life remaining. If necessary, define a replacement tool.

14.2 Compensation during program run

Application

During program run, you can open the active datum table and edit the values.

Related topics

- Contents and creation of a datum table
  Further information: Programming and Testing User's Manual
- Activating a datum table in the NC program
  Further information: Programming and Testing User's Manual

Description of function

The control opens the selected tables in the **Tables** operating mode.

The changed values only become effective after activating datum again.

14.2.1 Opening tables from within the Program Run operating mode

To open the compensation tables from within the **Program Run** operating mode:

Compensation tables
- ▶ Select **Compensation tables**
- > The control displays a selection menu.
- ▶ Select the table
  - **D**: Datum table
- > The control opens the selected table in the **Tables** operating mode.

Notes

> ### *NOTICE*
>
> **Danger of collision!**
>
> The control does not consider the changes made to a datum table or compensation table until the values have been saved. You need to activate the datum or compensation value in the NC program again; otherwise, the control will continue using the previous values.
>
> - ▶ Make sure to confirm any changes made to the table immediately (e.g., by pressing the **ENT** key)
> - ▶ Activate the datum or compensation value in the NC program again
> - ▶ Carefully prove-out the NC program after changing the table values

- When opening a table in the **Program Run** operating mode, the control will display the **M** status in the table tab. This status means that this table is active for program run.
- The clipboard allows you to transfer axis positions from the position display to the datum table.
  Further information: "Status overview of the TNC bar", Page 99

14.3   The Retract application

Application

The **Retract** application allows you to disengage the tool from the workpiece after an interruption in power (e.g., retraction of a tap engaged in the workpiece).

Requirement

- This application must be enabled by your machine manufacturer.

  The machine parameter **retractionMode** (no. 124101) allows the machine manufacturer to define whether the control will display the **Retract** toggle switch during start-up.

Description of function

The **Retract** application provides the following workspaces:

- **Retract**

  Further information: "The Retract workspace", Page 253

- **Positions**

  Further information: "The Positions workspace", Page 95

- **Status**

  Further information: "The Status workspace", Page 101

The **Retract** application provides the following buttons in the function bar:

Button	Meaning
Retract	Retract the tool with the axis keys or the electronic handwheel
End retraction	Close the **Retract** application The control opens the **End retraction?** window and prompts you to answer a confirmation request.
Start values	Reset the entries in the **A**, **B**, **C**, and **Thread pitch** fields to their original values

You select the **Retract** application by using the **Retract** toggle switch if the following conditions apply during start-up:

- Power interrupted
- No control voltage for the relay
- The **Move to ref. point** application

If you have activated a feed rate limit before the power failure occurred, this feed rate limit will still be active. When you select the **Retract** button, the control will display a pop-up window: This window allows you to deactivate the feed rate limit.

Further information: "Feed rate limit F LIMIT", Page 234

The Retract workspace

The **Retract** workspace provides the following contents:

Row	Meaning
Traversing mode	Traverse mode for retraction: ■ **Machine axes**: Move in the machine coordinate system **M-CS** ■ **Tilted system**: Without function for TNC7 go ■ **Tool axis**: Without function for TNC7 go ■ **Thread**: Move in the tool coordinate system **T-CS** with compensating movements of the spindle **Further information:** "Reference systems", Page 172
Kinematics	Name of the active machine kinematics
A, **B**, **C**	Current position of the rotary axes Effective in the **Tilted system** traverse mode Without function for TNC7 go
Thread pitch	Thread pitch from the **PITCH** column of tool management Effective in the **Thread** traverse mode
Direct. of rotation	Direction of rotation of the thread-turning tool: ■ **Right-hand thread** ■ **Left-hand thread** Effective in the **Thread** traverse mode
Coordinate system for handwheel superimposition	Coordinate system in which handwheel superimpositioning takes effect Effective in the **Tool axis** traverse mode Without function for TNC7 go

The control selects the mode of traverse and the associated parameters automatically. If the traverse mode or the parameters have not been correctly preselected, you are able to reset them manually.

Notes

> **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

■ Use the **+**, **-**, *****, **/**, **(**, and **)** keys for calculations in the numerical input fields.

Example

The power failed while a thread cutting cycle 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.

> In the **Start** workspace, the control provides the **Power interrupted** column.

▶ Activate the **Retract** toggle switch
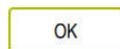
▶ Select **OK**

> The control compiles the PLC program.

▶ Switch the machine control voltage on

> The control checks the functioning of the emergency stop circuit

> The control opens the **Retract** application and displays the **Assume position values?** window.

▶ Compare the displayed position values with the actual position values
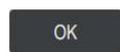
▶ Select **OK**

> The control closes the **Assume position values?** window

▶ Select the **Thread** traverse mode as needed

▶ Enter the thread pitch as needed

▶ Enter the direction of rotation as needed

▶ Select **Retract**

▶ Retract the tool with the axis keys or the handwheel

▶ Select **End retraction**

> The control opens the **End retraction?** window and prompts you to answer a confirmation request.
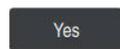
▶ If the tool was correctly retracted, select **Yes**

> The control closes the **End retraction?** window and the **Retract** application.

15

Tables

15.1 The Tables operating mode

Application

In the **Tables** operating mode you can open various tables and edit them as necessary.

Description of function

If you select **Add**, the control displays the **Quick selection new table** and **Open File** workspaces.

In the **Quick selection new table** workspace you can create a new table and open some tables directly.

Further information: Programming and Testing User's Manual

In the **Open File** workspace, you can open an existing table or create a new table.

Further information: Programming and Testing User's Manual

Multiple tables can be open at the same time. The control displays each table in a separate workspace.

If a table is selected for program run or simulation, the control shows the status **M** or **S** on the tab of the application. The status of the active application is highlighted in color and for the remaining applications in gray.

You can open the **Table**, **Form and Document** workspaces in every application.

Further information: "The Table workspace", Page 262

Further information: "The Form workspace for tables", Page 270

You can select various functions by using the context menu (e.g., **Copy**).

Further information: Programming and Testing User's Manual

> ⚙ Refer to your machine manual.
>
> This User's Manual describes the basic functions of the control. The machine manufacturer can adapt, enhance or restrict the control functions to the machine. The machine manufacturer can also change the colors in the control's GUI.

Buttons

In the **Tables** operating mode, the function bar contains the following buttons that can be used for any table:

Button	Meaning
Undo	The control undoes the last change.
Redo	The control restores the change that was undone.
GOTO record	The control opens the **GOTO jump instruction** window. The control selects the line number that you defined.
Edit	If the toggle switch is active, you can edit the table.
Mark row	The control marks the currently selected row.

Depending on the selected table, the control provides the following additional buttons in the function bar:

Button	Meaning
Insert rows	The control opens the **Insert rows** window where you can insert one or more new rows.
	If you enable the **Append** checkbox, the control will insert the rows after the last table row.
Reset row	The control resets all data contained in the row.
Delete rows	The control deletes the currently selected row.
Insert tool	The control opens the **Insert tool** window where you can define the following:
	■ **Type:**
	Further information: "Tool types", Page 161
	■ **Line number (Tool number?)**
	■ **Number of rows**
	■ **Index**
	Further information: "Indexed tool", Page 157
	■ **Append**
	Append rows at the end of the table
	Further information: "Tool management ", Page 163
Delete tool	The control deletes the tool selected in the tool management.
	You cannot delete any tools that have been entered into the pocket table. The button is dimmed.
	Further information: "Tool management ", Page 163
Import	The control imports table contents from a CSV file (e.g., tool data).
	Further information: "Importing and exporting of table contents", Page 273
Inspect	The control inspects a tool.
Unload	The control unloads a tool.
Load	The controls loads a tool.
Activate the preset	The control activates the currently selected row of the preset table as preset.
	Further information: "Preset table *.pr", Page 295
Lock record	The control locks the currently selected row of the preset table and thus protects the contents from changes.
	Further information: "Write-protection for table rows", Page 296

⚙ Refer to your machine manual.
  If necessary, the machine manufacturer adapts the buttons.

If at least one table row is marked, the control contains the following buttons in the action bar:

- **Cut**
- **Copy**
- **Reset**
  Only for tables that also provide the **Reset row** button
- **Delete**
- **Select all**
- **Cancel marking**
- **Export**
  Export data of the selected lines as a CSV file
  Further information: "Importing and exporting of table contents", Page 273
- **Cancel**

15.1.1 Editing the contents of tables

To edit the contents of a table:

▶ Select the desired table cell

Edit

▶ Activate **Edit**
> The control enables the values for editing.

ⓘ To edit a table content, you can also double-tap or double-click the table cell. The control displays the **Editing disabled. Enable?** window. You can enable the values for editing or cancel the process.

ⓘ If the **Edit** toggle switch is enabled, you can edit the contents both in the **Table** workspace and in the **Form** workspace.

Notes

- Use the **+**, **-**, *****, **/**, **(**, and **)** keys for calculations in the numerical input fields.
- The control enables you to transfer tables from previous controls to the TNC7 go and to adapt them automatically, if needed.
- When you open a table where columns are missing, for example in case of a tool table from a previous control, the control will display the **Incomplete table layout** window.

  When you create a new table in the file manager, the table does not contain information on the required columns yet. When you open the table for the first time, the **Incomplete table layout** window will open in the **Tables** operating mode.

  In the **Incomplete table layout** window, a selection menu allows you to select a table template. The control shows which table columns are added or removed, if applicable.
- If you have, for example, processed tables in a text editor, the control provides the **Update TAB / PGM** function. Use this function to complete an incorrect table format.

ⓘ Edit tables only by using the table editor in the **Tables** operating mode to avoid errors (e.g., format errors).

Notes about machine parameters

Refer to your machine manual.

- Using the optional machine parameter **CfgTableCellCheck** (no. 141300), the machine manufacturer can define rules for table columns. The machine parameter allows you to define columns as mandatory fields or reset them automatically to a default value. If this rule is not fulfilled, the control will display an information symbol.
- The machine manufacturer uses the machine parameter **CfgTableCellLock** (no. 135600) to define in which cases individual table cells will be blocked or write-protected. On some machines, you cannot change the tool type once a tool has been inserted into the machine.

15.2 The Create new table window

Application

You can create tables using the **Create new table** window in the **Quick selection new table** workspace.

Related topics

■ The **Quick selection new table** workspace

Further information: Programming and Testing User's Manual

■ Available file types for tables

Further information: Programming and Testing User's Manual

Description of function
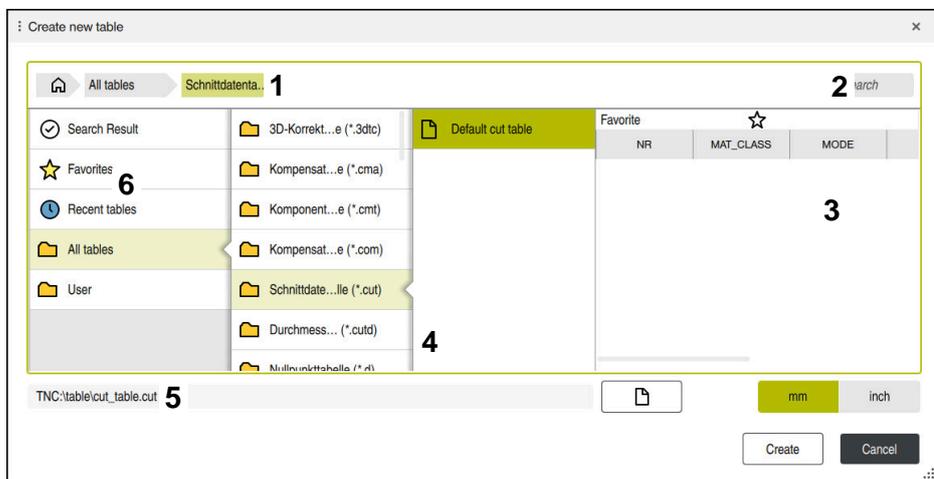
The **Create new table** window

The **Create new table** window shows the following areas:

1 Navigation path

In the navigation path the control shows the position of the current folder in the folder structure. Use the individual elements of the navigation path to move to a higher folder level. You can change the path or use the history to open a previous path.

2 Content columns

The control shows a folder and the available prototypes for each table type.

3 Searching

You can search for any strings. The control displays the results under **Search Result**.

4 The control shows the following information and functions:

■ Add or remove a favorite

■ Preview

5 Unit of measure (mm or inches)

6 Path of the table to be created

7   Navigation column

The navigation column offers the following possibilities for navigation:

- **Search Result**
- **Favorites**

  The control displays all folders and prototypes that you have marked as favorites.

- **Last functions**

  The control shows the eleven most recently used prototypes.

- **All functions**

  The control shows all available table types in the folder structure.

Notes

- 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 data are input or read.
- With the optional machine parameter **CfgTableCreate** (no. 140900), the machine manufacturer can provide additional areas in the navigation column (e.g., tables for the user).
- With the optional machine parameter **dialogText** (no. 105506), the machine manufacturer can define other names for the table types (e.g., tool table instead of **t**).

15.3 The Table workspace

Application

In the **Table** workspace, the control shows the contents of a table. You can search in all tables and filter the table content.

Description of function

The **Table** workspace

In the **Tables** operating mode, the **Table** workspace is open in every application by default.

The control displays the name and path of the file above the header of the table.

When you select the title of a column, the control will sort the table contents by this column in ascending or descending order.

If the table allows it, you can also edit the table contents in this workspace.

If you select a cell, the control displays the following information on the selected parameter in the dialog bar:

- Name
- Text width or input range
- Unit of measure as needed

⚙ Refer to your machine manual.

If necessary, the machine manufacturer adapts the contents displayed (e.g., the titles of table columns).

Icons and shortcuts

The **Table** workspace contains the following icons or shortcuts:

Icon or shortcut	Meaning
☰	Open or close the **Filter** column **Further information:** "The Filter column in the Table workspace", Page 264
🔍 **[CTRL] + [F]**	Open or close the **Search** column **Further information:** "The Search column in the Table workspace", Page 266
⚠	▪ Activate or deactivate **"Rules not met" filter** The control displays only the rows that do not meet the rules defined by the machine manufacturer in **CfgTableCellCheck** (no. 141300). ▪ Open the **Data record consistency violations** window In these rows, the control shows the icon at the start of the row, even if the filter is inactive. The control displays a window showing how the row violates the rules.
🖊	**Edit table characteristics** **Further information:** Programming and Testing User's Manual
100%	Current size of the content Open or close the **Scale** selection menu
🔍	**Reset scaling** Set the font size of the table to 100%
⚙	Open or close settings in the **Tables** window **Further information:** "Settings in the Table workspace", Page 267
✎	Open the **Search** column and edit the selected filter Only in the **Filter** column **Further information:** "The Search column in the Table workspace", Page 266
🗑	Delete the selected filter Only in the **Filter** column **Further information:** "The Filter column in the Table workspace", Page 264
[CTRL] + [A]	Mark all rows
[CTRL] + [SPACE]	Mark the active row or end the marking function
[SHIFT] + [UP]	Additionally mark the row above
[SHIFT] + [DOWN]	Additionally mark the row below

The Filter column in the Table workspace

The control provides default filters for the following tables:

- **Tool management**
- **Pocket table**
- **Presets**
- **Tool table**

For all other tables, you can create user-defined filters.

Further information: "User-defined filters", Page 266

The **Filter** column in the **Table** workspace is divided into several filter groups. The control separates the filter groups with a white double line.
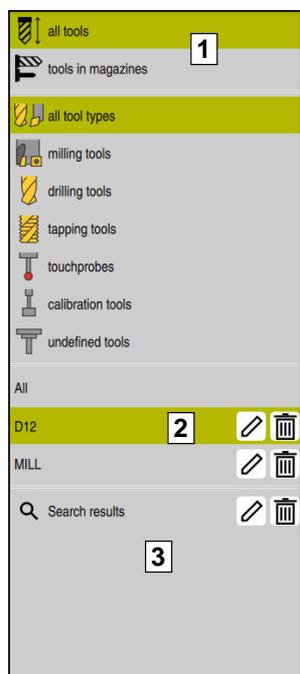
The control provides the following filter groups:

1 Default filters

Filters that are available by default for the respective table

In the **Tool management** application, the default filters comprise two filter groups

2 User-defined filters

Search processes saved as filters

Further information: "User-defined filters", Page 266

3 **Search results**

Results of the **Search** column

Further information: "The Search column in the Table workspace", Page 266

When you tap or click a filter once, the control activates only the selected filter in the respective group.

When you double-tap or click a filter, the control activates the selected filter in addition to the active filters.

Further information: "Connecting conditions and filters", Page 266

The control highlights active filters green.

⚙ Refer to your machine manual.
This User's Manual describes the basic functions of the control. The machine manufacturer can adapt, enhance or restrict the control functions to the machine. The machine manufacturer can also change the colors in the control's GUI.

Filters in the Tool management

The control provides the following default filters in the **Tool management**:

- **All tools**
- **Magazine tools**
- **All types**
- **Milling cutters**
- **Drills**
- **Taps**
- **Thread cutters**
- **Touch probes** (#17 / #1-05-1)
- **Calibration tools**
- **Undefined tools**

Filters in the Pocket table

The control provides the following default filters in the **Pocket table**:

- **All magazines**
- **spindle**
- **main magazine**
- **all pockets**
- **empty pockets**
- **occupied pockets**
- **Locked pockets**

Filters in the Presets table

The control provides the following default filters in the **Presets** table:

- **Show all**
- **Offsets**

User-defined filters

You can additionally create user-defined filters by saving a search.

Further information: "The Search column in the Table workspace", Page 266

The control only shows this filter group when you create a user-defined filter. In addition to the user-defined filters, the control displays the **All** filter option.

Further information: "The Filter column in the Table workspace", Page 264

Connecting conditions and filters

The control connects the filters as follows:

- AND operation for several requirements within one filter

  You create, for example, a user-defined filter that contains the requirements **R = 8** and **L > 150**. The control filters the table rows when you activate this filter. The control displays only the table rows that meet both requirements at the same time.

- OR operation between filters of the same filter groups

  When you activate the default filters **Drills** and **Taps**, for example, the control filters the table rows. The control displays only the table rows that meet at least one of the conditions. The table row must contain either a drill or a tap.

- AND operation between filters of different filter groups

  You create, for example, a user-defined filter that contains the requirement **R > 8**. When you activate this filter and the default filter **Milling cutters**, the control filters the table rows. The control displays only the table rows that meet both requirements at the same time.

The Search column in the Table workspace

The control offers a search function in all tables.

You can define multiple search conditions in the **Search** function.

Each condition includes the following information:

- Table column, such as **T** or **NAME**

  Use the **Search in** selection menu to select the column.

- Operator if applicable (e.g., **Contains** or **Equal to (=)**)

  Use the **Operator** selection menu to select the operator.

- Search term in the **Search for** input field

> **i** If you search the columns using predefined selection values, the control offers a selection menu instead of the input field.

The control provides the following buttons:

Button	Meaning
+	Use **Add** to add several conditions. The conditions will have a combined effect when you perform the search. You can save several conditions in a user-defined filter.
Search	The control searches the table.
Reset	The control resets the entered conditions and removes any additional conditions.
Save	You can save the entered conditions as a user-defined filter. You can assign any name to the filter. **Further information:** "User-defined filters", Page 266

An unsaved search works like a user-defined filter. If an unsaved search is active, the control highlights the **Search results** filter group green in the **Filter** column.

Further information: "The Filter column in the Table workspace", Page 264

> ⚙ Refer to your machine manual.
> This User's Manual describes the basic functions of the control. The machine manufacturer can adapt, enhance or restrict the control functions to the machine. The machine manufacturer can also change the colors in the control's GUI.

Settings in the Table workspace

In the **Tables** window, you can influence the contents shown in the **Table** workspace.

The **Tables** window consists of the following areas:

- **General**
- **Column sequence**

The General area

The setting selected in the **General** area is modally effective.

If the **Synchronize table and form** toggle switch is active, the cursor will move synchronously. If, for example, you select a different table column in the **Table** workspace, the control moves the cursor synchronously in the **Form** workspace.

The Column sequence area
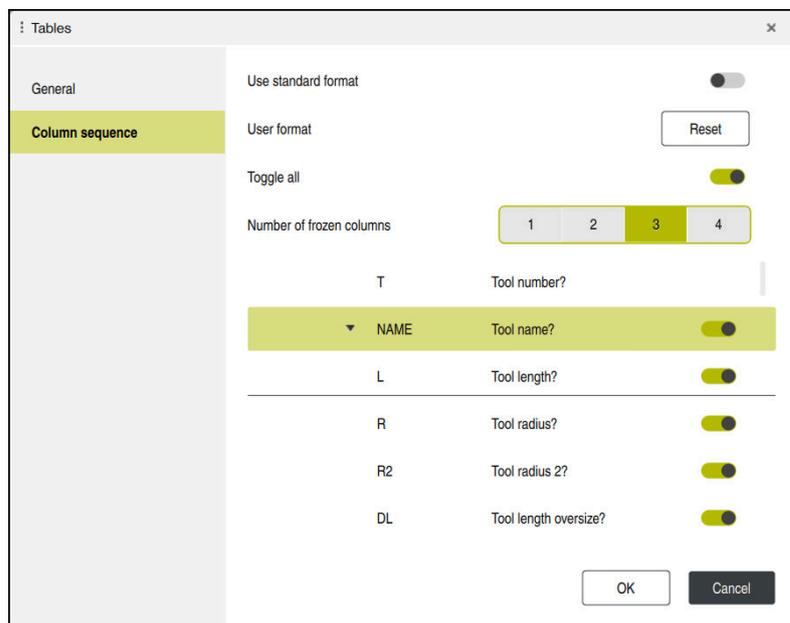
The **Tables** window

The **Column sequence** area contains the following settings:

Setting	Meaning
Use standard format	If you activate the toggle switch, the control shows all table columns, indicating them in the standard sequence.
	If you deactivate the toggle switch, the control restores the previous setting.
User format	If you select the **Reset** button, the control resets the adaptations to the settings of the standard format.
Toggle all	If you activate the toggle switch, the control shows all table columns.
	If you deactivate the toggle switch, the control hides all table columns.
	The first column in each table cannot be hidden.
Number of frozen columns	You define how many table columns the control freezes at the left table edge. You can freeze up to four table columns.
	These table columns will remain visible even when you navigate further to the right within the table.
Columns of the currently opened table	The control displays all table columns below each other. Use the toggle switches to separately hide or show each table column.
	The control displays a line below the selected number of frozen columns.
	When you select a table column, the control displays up and down arrows. Use these arrows to change the sequence of the columns.
	The respective first column in the table cannot be shifted.

The settings in the **Column sequence** area only apply to the currently opened table.

Virtual columns

In the **Table** workspace, the control can display virtual columns. Virtual columns are not included in the table files but contain calculated values from other data.

The control contains the following virtual columns:

Column	Meaning	Application
MAGAZIN	The control shows whether the tool is currently in the magazine or in the spindle.	■ Tool management ■ Pocket table
TOOL_LIFE	Information on the tool life: ■ **+**: The current tool life **CUR_TIME** is at least 5 min below the maximum tool life **TIME2**. ■ **-**: The current tool life **CUR_TIME** is less than 5 min below the maximum tool life **TIME2**. ■ **X**: The tool has reached the maximum tool life **TIME2**. ■ **?**: No value defined for the maximum tool life **TIME2**	■ Tool management ■ Pocket table

> ⚙ Refer to your machine manual.
> The machine manufacturer can modify virtual columns and define further columns.

Notes

- If you insert the contents of the clipboard while editing is inactive, the control displays the **Editing disabled. Enable?** prompt. If you select **Yes**, the control inserts the copied contents.
- Refer to your machine manual.
  Using the optional machine parameter **freeInputEnabled** (no. 105706), the machine manufacturer specifies whether you can enter custom parameters in selection menus.
- Since the values in virtual columns are not included in the table file, you cannot read these values with functions such as:
  - NC functions (e.g., **TABDATA READ**)
  - **HEIDENHAIN DNC** (#18 / #3-03-1)
  - **OPC UA NC Server** (#56-61 / #3-02-1*)

15.4 The Form workspace for tables

Application

In the **Form** workspace, the control shows all contents of a selected table row. Depending on the table, you can edit the values in the form.

Description of function
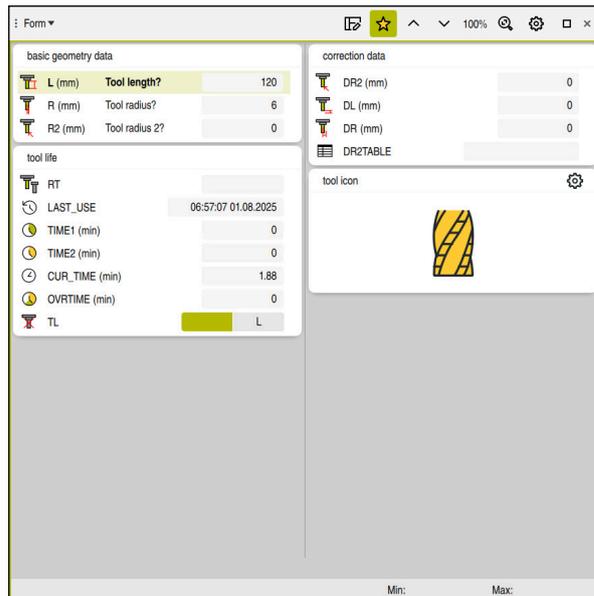
The **Form** workspace in the **Favorites** view

The control displays the following information for each parameter:

- An icon, if applicable
- Name
- Text width or input range
  Only in the dialog bar
- Unit of measure as needed
  Only in the dialog bar
- Description
- Current value

The control displays the contents of specific tables in groups within the **Form** workspace.

> Refer to your machine manual.
>
> If necessary, the machine manufacturer adapts the contents displayed (e.g., the titles of table columns).

Icons

The **Form** workspace contains the following icons or shortcuts:

Icon or shortcut	Meaning
🗒	**Configure the layout** You can make the following layout adaptations: ■ Add or remove areas to the **Favorites** view ■ Rearrange areas using the gripper ■ Add or remove columns
⭐	**Favorites** In this view, the control shows the areas that are marked as favorites. You can create a user-defined view using the favorites. If the icon is not active, the control shows all areas.
⌃ [SHIFT] + [UP] ⌄ [SHIFT] + [DOWN]	**Navigate** Navigate between table rows
100%	Current size of the content Open or close the **Scale** selection menu
🔍	**Reset scaling** Set font size of the content to 100%
⚙	**Settings** ■ Open the settings in the **Tables** window **Further information:** "Settings in the Form workspace", Page 272 ■ Change the size of the graphic in the **Tool Icon** area
＋	**Add** The control only shows this icon when you are adapting the layout. With this icon you can add the following elements: ■ Column You can divide the workspace into several columns. **Further information:** "Adding a column in the workspace", Page 272 ■ Area In the **Favorites** view you can add another area.
▬	**Remove** The control only shows this icon when you are adapting the layout. You can delete an empty column with this icon.

Settings in the Form workspace

In the **Tables** window, you can select whether the control will show the parameter descriptions. The selected setting is modally effective.

Tables	×
General	Show column descriptions ⬤

OK   Cancel
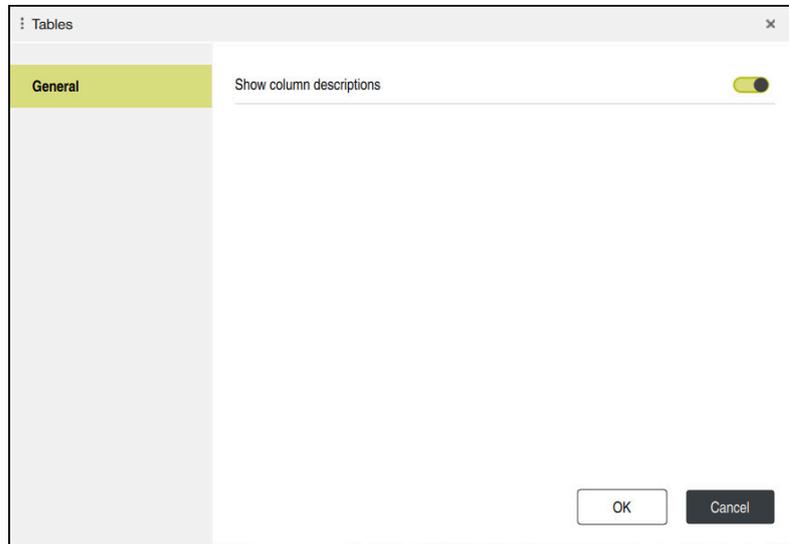
15.4.1 Adding a column in the workspace

To add a column:

⬚ ▶ Select **Configure the layout**
  > The control enables all functions for adapting the layout of the workspace.
  ▶ In the workspace, swipe to the left

＋ ▶ Select **Add**
  > The control adds a new column.

⋮ ▶ Move the areas if required

⬚ ▶ Select **Configure the layout**
  > The control saves your changes.

Notes

- In the tool manager, the control displays the relevant parameters for the current tool type only and hides all other parameters. If you select the **Configure the layout** icon, the control will also display the hidden form areas.
- The control displays an icon of the selected tool type in the **Tool Icon** area.
  Further information: "Tool types", Page 161
- Refer to your machine manual.
  Using the optional machine parameter **freeInputEnabled** (no. 105706), the machine manufacturer specifies whether you can enter custom parameters in selection menus.

15.5 Importing and exporting of table contents

Application

You can import the contents of specific tables into the control and export them from the control. This avoids manual editing efforts and possible typing errors.

Importing tool data is particularly useful in connection with a tool presetter. Via exported contents you can, for example, transfer the tools and presets of the control to the CAM system.

The control uses CSV files for exporting and importing of table contents.

Related topics

- Overview of file types

  Further information: Programming and Testing User's Manual

- Fundamentals of the **Tool management** application

  Further information: "Tool management ", Page 163

Description of function

Importing of data is possible with all tables that contain the **Import** button in the function bar.

To export contents, you have to mark all desired table rows. If a table does not permit exporting, the **Export** button is grayed out in the action bar.

The CSV file has the following structure:

- The first line contains the column names of the table being exported.
- The other lines contain the exported contents. The contents are arranged in the same sequence as the first line. Decimal numbers are separated by a point.

For compatibility with earlier controls, the control provides the following formatting options for exporting:

- **TNC7 (semicolon-separated)** writes the values between double quotation marks and separates the values by semicolons.

  Most table calculation programs use this formatting.

- **iTNC 530 / TNC 640 (comma-separated)** writes the values partly between brackets and separates the values by commas

The control is able to import and export data in both formats.

The Confirm import window

When you select a CSV file for importing, the control will display the **Confirm import** window.
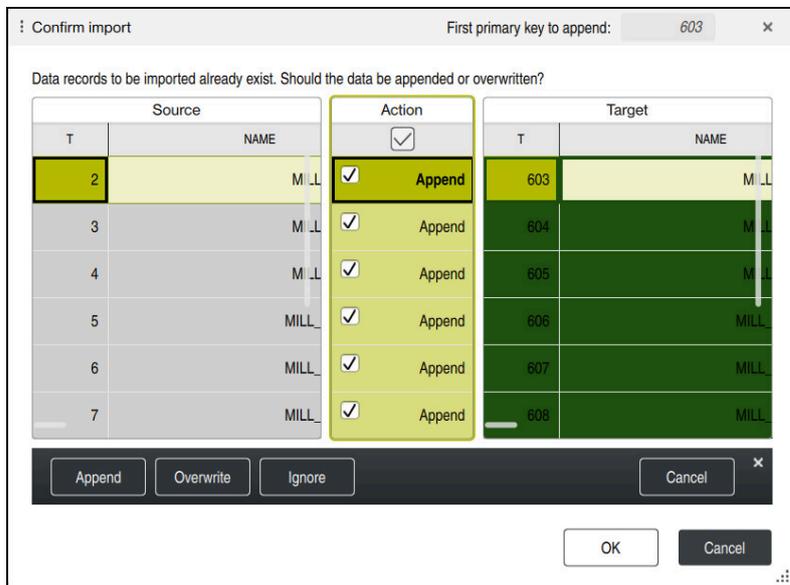
The **Confirm import** window with table rows to be appended

The window contains the following areas:

Area	Meaning
Source	The control shows the values of the CSV file that are to be imported.

Action	If you double-tap or double-click a row, the following actions for this row can be selected:
	
	■ **Append**
	  Append row at end of table
	■ **Overwrite**
	  Import row with line number of the CSV file
	■ **Ignore**
	  Do not import row of the CSV file
	
	Clicking the checkbox in the header selects all table rows. You can select the desired action for all rows that have been marked in the action bar.

Target	The control shows the table values that will result from the currently selected action after importing. Depending on the selected action, the control shows the cells in the following colors:

Color	Meaning	Action
Yellow	The content of the source differs from the content of the target table.	**Ignore**
Dark gray	The table will not contain a row with this number after importing.	**Ignore**
Dark green	The rows will be added in the target file.	**Append**
Red	The current table content will be overwritten after importing.	**Overwrite**

The control displays an input field in the title bar of the window. If **Append** is selected, you can optionally enter a row number that will be the start number of the imported table rows. This allows you to define new tool numbers, for example.

15.5.1 Exporting table contents

To export table contents:

⊞ ▶ Select the **Tables** operating mode

▶ Select the table (e.g., **Tool management**)

▶ Select the row to be exported

Mark row ▶ Select **Mark row**

> The control marks the selected line and opens the action bar.

▶ Select further rows, if required
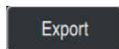
Export ▶ Select **Export**

> The control opens the **Save as** window.

▶ Select a path

▶ Enter the file name

▶ Select the formatting

Save ▶ Select **Save**

> The control saves the CSV file using the defined path.

15.5.2 Importing table contents

To import table contents:

▶ Select the **Tables** operating mode

▶ Select the table (e.g., **Tool management**)

▶ Activate **Edit**

> The control enables the table for editing.

▶ Select **Import**

> The control opens a selection window.

▶ Select the desired CSV file

▶ Select **Import**

> The control opens the **Confirm import** window.

▶ Click the checkbox in the header of the **Action** area

> The control marks all rows to be imported and opens the action bar.

▶ Select an action:

- **Append**: the control adds the contents as new rows at the end of the table.
- **Overwrite**: the control overwrites the initial contents with the contents from the CSV file.
- **Ignore**: the control does not import the contents.

▶ Select **OK**

> The control imports the contents with the selected action.

ℹ If you double-tap or double-click each cell in the **Action** area, the following actions can be selected separately for each row.

NOTICE

Caution: Data may be lost!

When overwriting existing table values with the **Overwrite** function, the control will permanently delete the initial values!

▶ Use this function only with table files that are no longer needed

Notes

<table>
<tr><td colspan="2" align="center">**NOTICE**</td></tr>
<tr><td colspan="2">**Caution: Possible material damage!**</td></tr>
<tr><td colspan="2">If the transfer file contains unknown column names, the control will not accept the data from this column! In this case, the control will perform the operations with an incompletely defined tool.</td></tr>
<tr><td>▶</td><td>Check whether the column names are correct</td></tr>
<tr><td>▶</td><td>After importing, check the tool data and adapt them if required.</td></tr>
</table>

- The CSV file contains information on the table that the export comes from. Only tables of the same file type allow importing.
- By default, the control saves the export under **TNC:\system\tooltab**.
- To be able to import the CSV file, the column with the line numbers must be available.
- In the **Presets** application, the **Confirm import** window does not include an input field in the title bar. Within the preset table, you can append rows only using the next line number.
- If rows cannot be overwritten when importing CSV files, the control will display an information symbol. If you select the information symbol, the control displays a pop-up window showing the reason for the problem. The control provides a button with the information symbol. It allows you to filter the contents to find all rows with this problem.

15.6 Tool tables

15.6.1 Overview

This chapter contains information about the tool tables of the control.

■ Tool table **tool.t**

   Further information: "Tool table tool.t", Page 278
■ Touch probe table **tchprobe.tp** (#17 / #1-05-1)

   Further information: "Touch probe table tchprobe.tp (#17 / #1-05-1)", Page 285

You can edit the tools, except for the touch probes, in tool management only.

Further information: "Tool management ", Page 163

15.6.2 Tool table **tool.t**

Application

The tool table **tool.t** contains the parameters specific to drilling and milling tools. The tool table also contains all parameters that are independent of the technology, such as the tool life **CUR_TIME**.

Related topics

■ Editing parameters in tool management

   Further information: "Tool management ", Page 163
■ Tool parameters

   Further information: "Tool parameters", Page 154

Description of function

The file name of the tool table is **tool.t** and this table must be stored in the folder **TNC:\table**.

Parameters of the tool table tool.t

The **Tool_management** application shows the following parameters of the tool table:

> **i** The **tool.t** file contains, if applicable, additional table columns that are not shown in the **Tool_management** application. These additional columns are not relevant for the TNC7 go.

Parameter	Meaning
T	**Tool number?**
	Row number in the tool table
	The tool number allows you to identify each tool unambiguously (e.g., for calling a tool).
	Further information: Programming and Testing User's Manual
	You can define an index after the period.
	Further information: "Indexed tool", Page 157
	This parameter applies to all tools, regardless of technology.
NAME	**Tool name?**
	The tool name identifies a tool, for example when calling it.
	Further information: Programming and Testing User's Manual
	You can define an index after a period (i.e., name.index).
	Further information: "Indexed tool", Page 157
	This parameter applies to all tools, regardless of technology.
L	**Tool length?**
	Length of tool, with respect to the tool carrier reference point
	Further information: "Tool carrier reference point", Page 151
R	**Tool radius?**
	Tool radius, with respect to the tool carrier reference point
	Further information: "Tool carrier reference point", Page 151
R2	**Tool radius 2?**
	Corner radius for the exact definition of the tool for graphic representation and collision monitoring of, for example, ball-nose cutters or toroid cutters.
DL	**Tool length oversize?**
	Delta value of the tool length as the compensation value
	Is added to the parameter **L**
DR	**Tool radius oversize?**
	Delta value of the tool radius as the compensation value
	Is added to parameter **R**
DR2	**Tool radius oversize 2?**
	Delta value of the tool radius 2 as the compensation value
	Is added to parameter **R2**

Parameter	Meaning
TL	**Tool locked?**
	Tool is enabled or locked for machining:
	■ No value entered: Enabled
	■ **L**: Locked
	The control locks the tool after exceeding maximum tool age **TIME1**, maximum tool age 2 **TIME2** or after exceeding one of the parameters for automatic tool measurement.
	This parameter applies to all tools, regardless of technology.
RT	**Replacement tool?**
	Number of the replacement tool
	If the control calls a tool in a TOOL CALL and the tool is not available or locked, the control inserts the replacement tool.
	If the replacement tool is not available or locked, the control inserts the replacement tool of the replacement tool.
	You can define an index after the period.
	Further information: "Indexed tool", Page 157
	If you define the value 0, the control will not use a replacement tool.
	This parameter applies to all tools, regardless of technology.
	Selection by means of a selection window
TIME1	**Maximum tool age?**
	Maximum tool age in minutes
	If the current tool age **CUR_TIME** exceeds the **TIME1** value, the control locks the tool and displays an error message when the tool is called the next time.
	The behavior depends on the machine. Refer to your machine manual.
	This parameter applies to all tools, regardless of technology.
TIME2	**Max. tool age for TOOL CALL?**
	Maximum tool age 2 in minutes
	The control inserts a replacement tool in the cases below:
	■ When the current tool age **CUR_TIME** exceeds the **TIME2** value, the control locks the tool. The control no longer inserts the tool when the tool is called. If a replacement tool **RT** is defined and available in the magazine, the control inserts the replacement tool. If no replacement tool is available, the control will display an error message.
	Further information: Programming and Testing User's Manual
	The behavior depends on the machine. Refer to your machine manual.
	This parameter applies to all tools, regardless of technology.
CUR_TIME	**Current tool age?**
	The current tool age equals the time during which the tool is cutting a workpiece. The tool is cutting a workpiece when the spindle is switched on and the control moves the tool at the machining feed rate. The control counts this time automatically and enters the current tool age in minutes.
	You can edit the tool age of an active tool during program run after you have inserted an indexable insert, for example. The control will directly apply the value to tool life monitoring.
	The control updates the value cyclically during NC program run, as well as during a tool call and at the end of the program.
	This parameter applies to all tools, regardless of technology.

Parameter	Meaning
TYP	**Tool type?**
	Depending on the selected tool type, the control displays the suitable parameters in the **Form** workspace of the tool management.
	Further information: "Tool types", Page 161
	Further information: "Tool management ", Page 163
	This parameter applies to all tools, regardless of technology.
	Selection using a selection menu
DB_ID	**ID for central tool management**
	The database-ID allows you to identify a tool (e.g., within a tool management system by using client applications).
	Further information: "Database ID", Page 156
	For indexed tools, HEIDENHAIN recommends that you assign the database ID to the main tool.
	Further information: "Indexed tool", Page 157
	This parameter applies to all tools, regardless of technology.
DOC	**Tool description**
	This parameter applies to all tools, regardless of technology.
PLC	**PLC status?**
	Tool information for the PLC
	Refer to your machine manual.
	This parameter applies to all tools, regardless of technology.
LCUTS	**Tooth length in the tool axis?**
	Length of cutting edge for exact definition of the tool for graphical simulation, automatic calculation within cycles and collision monitoring.
LU	**Usable length of the tool?**
	Usable length of the tool for exact definition of the tool for graphical simulation, automatic calculation within cycles and collision monitoring (e.g., of necks of end mills).
RN	**Neck radius of the tool?**
	Neck radius for the exact definition of the tool for graphic simulation and collision monitoring of, for example, necks of end mills or side milling cutters.
	The tool can contain a neck radius **RN** only if the useful length **LU** is longer than the **LCUTS** length of the cutting edge.
R_TIP	**Radius at the tip**
	Radius at the tool tip for exact definition of the tool for graphical simulation and automatic calculation within cycles for tools such as countersinks.
ANGLE	**Maximum plunge angle?**
	Maximum plunge angle of the tool for reciprocating plunge-cutting in the cycles.
CUT	**Number of teeth?**
	Number of teeth of the tool for automatic tool measurement or cutting data calculation.
	Further information: Programming and Testing User's Manual
	Further information: Programming and Testing User's Manual

Parameter	Meaning
TMAT	**Tool material?**
	Tool material from the tool material table **TMAT.tab** for cutting data calculation.
	Further information: Programming and Testing User's Manual
	Selection by means of a selection window
CUTDATA	**Cutting data table?**
	Select the cutting data table with the ***.cut** or ***.cutd** file extension for cutting data calculation.
	Further information: Programming and Testing User's Manual
	Selection by means of a selection window
LTOL	**Wear tolerance: length?**
	Permitted tool length deviation in wear detection for automatic tool measurement.
	Further information: Programming and Testing User's Manual
	If the entered value is exceeded, the control locks the tool in the **TL** parameter.
RTOL	**Wear tolerance: radius?**
	Permitted tool radius deviation in wear detection for automatic tool measurement.
	Further information: Programming and Testing User's Manual
	If the entered value is exceeded, the control locks the tool in the **TL** parameter.
R2TOL	**Wear tolerance: Radius 2?**
	Permitted tool radius 2 deviation in wear detection for automatic tool measurement.
	Further information: Programming and Testing User's Manual
	If the entered value is exceeded, the control locks the tool in the **TL** parameter.
DIRECT	**Cutting direction?**
	Cutting direction for exact definition of a tool for graphical representation, automatic tool measurement, and calculation of traverse movements.
	For milling tools you indicate the rotational direction of the tool spindle with which the tool cuts.
	■ -: **M3**
	■ +: **M4**
R-OFFS	**Tool offset: radius?**
	Position of tool upon length measurement, offset between the center of the tool touch probe and the tool center for automatic tool measurement.
	Further information: Programming and Testing User's Manual
L-OFFS	**Tool offset: length?**
	Position of tool upon radius measurement, distance between the top edge of the tool touch probe and the tool tip for automatic tool measurement.
	Further information: Programming and Testing User's Manual
	Is added to the machine parameter **offsetToolAxis** (no. 122707)
LBREAK	**Breakage tolerance: length?**
	Permitted tool length deviation in breakage detection for automatic tool measurement.
	Further information: Programming and Testing User's Manual
	If the entered value is exceeded, the control locks the tool in the **TL** parameter.

Parameter	Meaning
RBREAK	**Breakage tolerance: radius?**
	Permitted tool radius deviation in breakage detection for automatic tool measurement.
	Further information: Programming and Testing User's Manual
	If the entered value is exceeded, the control locks the tool in the **TL** parameter.
NMAX	**Maximum speed [rpm]**
	Limitation of spindle speed for the programmed value including control by the potentiometer.
LIFTOFF	**Lift-off allowed?**
	Allow automatic tool lift-off with active **M148** or **FUNCTION LIFTOFF**:
	■ **Y**: Activate **LIFTOFF**
	■ **N**: Deactivate **LIFTOFF**
TP_NO	**Number of the touch probe**
	Number of touch probe in the touch probe table **tchprobe.tp**
	Further information: "Touch probe table tchprobe.tp (#17 / #1-05-1)", Page 285
	Selection by means of a selection window
T-ANGLE	**Point angle**
	Point angle of the tool for exact definition of the tool for graphical simulation, automatic calculation within cycles and collision monitoring of drilling tools, for example.
	Further information: Programming and Testing User's Manual
PITCH	**Tool thread pitch?**
	Thread pitch of the tool for automatic calculations within cycles. A positive sign means a right-hand thread.
	Further information: Programming and Testing User's Manual
LAST_USE	**Date/time of last tool usage**
	The time at which the tool was last used
	The control updates the value cyclically during NC program run, as well as during a tool call and at the end of the program.
	This parameter applies to all tools, regardless of technology.
PTYP	**Tool type for pocket table?**
	Tool type for evaluation in the pocket table
	Further information: "Pocket table tool_p.tch", Page 290
	Refer to your machine manual.
	This parameter applies to all tools, regardless of technology.
KINEMATIC	**Tool-carrier kinematics**
	Assigning a tool carrier for exact definition of the tool for graphical simulation and collision monitoring.
	Further information: "Tool carrier management", Page 165
	Selection by means of a selection window
	This parameter applies to all tools, regardless of technology.

Parameter	Meaning
OVRTIME	**Tool life expired**
	Time in minutes during which the tool may be used beyond the tool life defined in the **TIME2** parameter.
	The machine manufacturer defines the function of this parameter. The machine manufacturer defines how the control will use the parameter when searching for tool names. Refer to your machine manual.
	This parameter applies to all tools, regardless of technology.
RCUTS	**Width of the indexable insert**
	Front-face width of cutting edge for exact definition of the tool for graphical simulation, automatic calculation within cycles and collision monitoring (e.g., for indexable inserts).

Notes

- Use the machine parameter **unitOfMeasure** (no. 101101) to define inches as the unit of measure. This does not automatically change the unit of measure in the tool table!

  Further information: "Creating a tool table in inches", Page 289

- If you want to archive tool tables or use them for simulation, save them with different file names and the corresponding file extension.
- The control shows delta values from the tool management graphically in the simulation. For delta values from the NC program or from compensation tables, the control changes only the position of the tool in the simulation.
- Assign unique tool names!

  If you define identical tool names for multiple tools, the control will look for the tool in the following sequence:
  - Tool that is in the spindle
  - Tool that is in the magazine

  > Refer to your machine manual.
  > If there are multiple magazines, the machine manufacturer can specify the search sequence of the tools in the magazines.

  - Tool that is defined in the tool table but is currently not in the magazine

  If the control, for example, finds multiple available tools in the tool magazine, it inserts the tool with the least remaining tool life.

- In the machine parameter **offsetToolAxis** (no. 122707), the machine manufacturer defines the distance between the upper edge of the tool touch probe and the tool tip.

  The parameter **L-OFFS** is added to this defined distance.

- In the machine parameter **zeroCutToolMeasure** (no. 122724), the machine manufacturer defines whether the control takes the parameter **R-OFFS** into account for automatic tool measurement.
- With the optional machine parameter **resetOnTypeChange** (no. 125304), you can define how the control will react when the tool type is changed. The machine manufacturer enables this parameter. If this machine parameter has been set to **TRUE** and you change the tool type, the control will first display a confirmation prompt and then reset all tool parameters.

15.6.3 Touch probe table tchprobe.tp (#17 / #1-05-1)

Application

In the touch probe table **tchprobe.tp** you define the parameters of the touch probe for the touching process, such as the probing feed rate. If you use several touch probes, you can save separate parameters for each touch probe.

Related topics

- Editing parameters in tool management
  Further information: "Tool management ", Page 163
- Tool parameters
  Further information: "Tool parameters", Page 154
- Touch probe functions
  Further information: "Touch probe functions in the Manual operating mode (#17 / #1-05-1)", Page 209
- Calibrating touch probe cycles for the tool touch probe
  Further information: Programming and Testing User's Manual
- Automatic touch probe cycles for the tool
  Further information: Programming and Testing User's Manual

Description of function

The file name of the touch probe table is **tchprobe.tp** and this table must be stored in the folder **TNC:\table**.

Parameters of the touch probe table tchprobe.tp

The touch probe table **tchprobe.tp** provides the following parameters:

Parameter	Meaning
NO	**Sequential number of touch probe**
	Enter this number in the **TP_NO** parameter of the tool management. The control links the data of the touch probe table to the tool management.
TYPE	**Selection of the touch probe?**
	🛈 The TS 642 touch probe makes the following values available: ■ **TS642-3**: The touch probe is activated by a conical switch. This mode is not supported. ■ **TS642-6**: The touch probe is activated by an infrared signal. Select this mode.
	Selection using a selection menu
STYLUS	**Shape of the stylus**
	■ **SIMPLE**: Straight stylus ■ **L-TYPE**: L-shaped stylus
	If you do not define the parameter, the control uses **SIMPLE**
	Selection using a selection menu
CAL_OF1	**TS center misalignmt. ref. axis? [mm]**
	Depending on the selection of the **STYLUS** parameter, this parameter has the following function: ■ **SIMPLE**: Offset of the touch probe axis to the spindle axis in the main axis ■ **L-TYPE**: Length of extension on an L-shaped stylus
	Required when **ON** is selected in parameter **TRACK**
	The control describes this value in connection with the calibration cycle.
CAL_OF2	**TS center misalignmt. aux. axis? [mm]**
	Offset of the touch probe axis to the spindle axis in the secondary axis
	Required when **ON** is selected in parameter **TRACK**
	The control describes this value in connection with the calibration cycle.
CAL_ANG	**Spindle angle for calibration?**
	Depending on the selection of the **STYLUS** parameter, this parameter has the following function: ■ **SIMPLE**: Prior to calibrating or probing, the control orients the touch probe with this spindle angle (if possible). ■ **L-TYPE**: The control orients the extension using the spindle angle.
	Prior to calibrating or probing, the control aligns the touch probe with the spindle orientation angle (if possible).
	Required when **ON** is selected in parameter **TRACK**
F	**Probing feed rate? [mm/min]**
	In the machine parameter **maxTouchFeed** (no. 122602), the machine manufacturer defines the maximum probing feed rate.
	If **F** is greater than the maximum probing feed rate, then the maximum probing feed rate will be used.

Parameter	Meaning
FMAX	**Rapid traverse in probing cycle? [mm/min]**
	Feed rate at which the control pre-positions the touch probe and positions it between the measuring points
DIST	**Maximum measuring range? [mm]**
	If the stylus is not deflected in a probing process within the defined value, the control will display an error message.
SET_UP	**Set-up clearance? [mm]**
	Distance of touch probe from the defined touch point when pre-positioning
	The smaller this value is, the more exactly you must define the touch point position. Safety clearances defined in the touch probe cycle are added to this value.
F_PREPOS	**Pre-position at rapid? ENT/NOENT**
	Speed for pre-positioning:
	■ **FMAX_PROBE**: Pre-position at the speed from **FMAX**
	■ **FMAX_MACHINE**: Pre-position at machine rapid traverse
	Selection using a selection menu
TRACK	**Probe oriented? Yes=ENT/No=NOENT**
	Orienting the infrared touch probe in each probing process:
	■ **ON**: The control orients the touch probe in the defined probing direction. In this way, the stylus is always deflected in the same direction, improving measuring accuracy.
	■ **OFF**: The control will not orient the touch probe.
	When selecting **L-TYPE** in the **STYLUS** parameter, **ON** must be selected.
	If you change the **TRACK** parameter, you must recalibrate the touch probe.
SERIAL	**Serial number?**
	The control automatically edits this parameter of touch probes with an EnDat interface.
REACTION	**Reaction? EMERGSTOP=ENT/NCSTOP=NOENT**
	As soon as touch probes with a collision protection adapter detect a collision, they react by resetting the ready signal.
	Reaction to resetting the ready signal:
	■ **NCSTOP**: Interrupting an NC program
	■ **EMERGSTOP**: Emergency stop, quick braking of the axes
	Selection using a selection menu

Editing the touch probe table

To edit the touch probe table:

▶ Select the **Tables** operating mode

▶ Select **Add**
> The control opens the **Quick selection** and **Open File** workspaces.
▶ Select the **tchprobe.tp** file in the **Open File** workspace
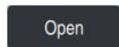
▶ Select **Open**
> The control opens the **Touch probes** application.

▶ Activate **Edit**
▶ Select the desired value
▶ Edit the value

Notes

- You can also edit the touch probe table values in the tool management.
- If you want to archive tool tables or use them for simulation, save them with different file names and the corresponding file extension.
- In the machine parameter **overrideForMeasure** (no. 122604), the machine manufacturer defines whether you will be allowed to change the feed rate with the feed-rate potentiometer during probing.

15.6.4 Creating a tool table in inches

To create a tool table in inches:

▶ Select the **Manual** operating mode

▶ Select **T**
▶ Select the tool **T0**
▶ Press the **NC Start** key
> The control removes the current tool and does not insert a new tool.
▶ Restart the control
▶ Do not acknowledge **Power interrupted**
▶ Select the **Files** operating mode

▶ Open the **TNC:\table** folder
▶ Rename the original file (e.g., **tool.t** as **tool_mm.t**)
▶ Select the **Tables** operating mode

▶ Select **Create new table**
> The control opens the **Create new table** window.
▶ Select a folder with the corresponding table type (e.g., **t**)
▶ Select INCH as the unit of measure if necessary
▶ Select the desired prototype

▶ Select a path
> The control opens the **Save as** window.
▶ Select the **table** folder
▶ Enter a name (e.g., **tool**)
▶ Select **Create** twice
> The control opens the **Tool table** tab in the **Tables** operating mode.
▶ Restart the control

▶ Acknowledge **Power interrupted** with the **CE** key

▶ Select the **Tool table** tab in the **Tables** operating mode
> The control uses the newly created table as a tool table.

ⓘ To use the **Tool management** application you have to create all existing tool tables in inches.

15.7 Pocket table tool_p.tch

Application

The **tool_p.tch** pocket table provides the pocket assignment of the tool magazine.
The control needs the pocket table in order to change the tool.

Related topics

- Tool call
  Further information: Programming and Testing User's Manual
- Tool table
  Further information: "Tool table tool.t", Page 278

Requirement

- The tool is defined in the tool management.
  Further information: "Tool management ", Page 163

Description of function

The file name of the pocket table is **tool_p.tch** and this table must be stored in the folder **TNC:\table**.

The **tool_p.tch** pocket table provides the following parameters:

Parameter	Meaning
P	**Pocket number?** Pocket number of the tool in the tool magazine
T	**Tool number?** Row number of the tool from the tool table With the machine parameter **deleteLoadedTool** (no. 125301) you define whether you are allowed to edit the **T** column. The machine manufacturer enables this parameter. **Further information:** "Tool table tool.t", Page 278 Selection by means of a selection window
RSV	**Reserve pocket?** When a tool is in the spindle, the control reserves the pocket of this tool in the box magazine. To reserve the pocket for the tool: ■ No value entered: Pocket is not reserved ■ **R**: Pocket is reserved
ST	**Special tool?** Define the tool as a special tool (e.g., with oversize tools): ■ No value entered: No special tool ■ **S**: Special tool
F	**Fixed pocket?** Always return the tool to the same pocket in the tool magazine (e.g., with special tools) To define a fixed pocket for the tool: ■ No value entered: No fixed pocket ■ **F**: Fixed pocket

Parameter	Meaning
L	**Locked pocket?** To lock a pocket for tools (e.g., the pockets next to special tools): ◼ No value entered: Do not lock ◼ **L**: Lock
NAME	**Tool name?** Name of the tool from the tool table When you define the tool number, the control will automatically load the tool name. **Further information:** "Tool table tool.t", Page 278
DOC	**Tool comment?** Tool comment from the tool table When you define the tool number, the control will automatically load the tool comment. **Further information:** "Tool table tool.t", Page 278
PLC	**PLC status?** Information about this tool pocket, which is transferred to the PLC The machine manufacturer defines the function of this parameter. Refer to your machine manual.
P1 ... P5	**Value?** The machine manufacturer defines the function of this parameter. Refer to your machine manual.
PTYP	**Tool type for pocket table?** Tool type for evaluation in the pocket table The machine manufacturer defines the function of this parameter. Refer to your machine manual.
LOCKED_ABOVE	**Lock pocket above?** Box magazine: Lock the pocket above This parameter depends on the machine. Refer to your machine manual.
LOCKED_BELOW	**Lock pocket below?** Box magazine: Lock the pocket below This parameter depends on the machine. Refer to your machine manual.
LOCKED_LEFT	**Lock pocket at left?** Box magazine: Lock the pocket at left This parameter depends on the machine. Refer to your machine manual.
LOCKED_RIGHT	**Lock pocket at right?** Box magazine: Lock the pocket at right This parameter depends on the machine. Refer to your machine manual.
S1	**S1** Value for evaluation in the PLC The machine manufacturer defines the function of this parameter. Refer to your machine manual.
S2	**S2** Value for evaluation in the PLC The machine manufacturer defines the function of this parameter. Refer to your machine manual.

Notes

- If a tool has been saved to the pocket table, you can neither reset the corresponding row in tool management nor delete the tool. The tool must be removed from the magazine first.
- Changing a tool parameter in the pocket table also changes that same parameter in the tool manager.
- In the **Table** workspace of the pocket table, the control displays the additional virtual **MAGAZINE** and **TOOL_LIFE** columns.

  Further information: "Virtual columns", Page 269

15.8 Tool usage file

Application

The control saves information about the tools of an NC program in a tool usage file (e.g., all the required tools and the tool usage times). The control needs this file for the tool usage test.

Related topics

- Using the tool usage test

    Further information: "Tool usage test", Page 169

    Further information: Programming and Testing User's Manual

- Tool data from the tool table

    Further information: "Tool table tool.t", Page 278

Requirements

- **Generate tool-usage file** is enabled by your machine manufacturer

    In the machine parameter **createUsageFile** (no. 118701), the machine manufacturer defines whether the **Generate tool-usage file** function will be enabled.

    Further information: "Creating the tool usage file", Page 169

- The **Generate tool-usage file** setting is set to **Once** or **Always**

    Further information: "The Machine and Simulation areas", Page 333

Description of function

The tool usage file provides the following parameters:

Parameter	Meaning
NR	Row number in the tool usage file
TOKEN	In the **TOKEN** column, the control uses one word to show which information is contained in the respective row:
	■ **TOOL**: Data per tool call; listed in chronological order
	■ **TTOTAL**: All data of a tool; listed in alphabetical order
	■ **STOTAL**: Called NC programs; listed in chronological order
	■ **TIMETOTAL**: Total tool usage time of an NC program
	■ **TOOLFILE**: Path of the tool table
	This enables the control during the tool usage test to detect whether you have performed the simulation with the tool table **tool.t**
TNR	Tool number
	If the control has not yet inserted a tool, the column contains the value **-1**.
IDX	Tool index
NAME	Tool name
TIME	Tool usage time in seconds
	Time during which the tool is cutting a workpiece (excluding rapid traverse movements)
WTIME	Total tool usage time in seconds
	Total time between the tool changes, during which the tool is cutting a workpiece

Parameter	Meaning
RAD	Sum of the tool radius **R** and the delta radius **DR** from the tool table
BLOCK	NC block number of the tool call
PATH	Path of the NC program or the tool table
T	Tool number, including the tool index If the control has not yet inserted a tool, the column contains the value **-1**.
OVRMAX	Maximum feed-rate override If you only simulate the machining operation, then the control will enter the value **100**.
OVRMIN	Minimum feed rate override If you only simulate the machining operation, then the control will enter the value **-1**.
NAMEPRG	Type of tool definition during a tool call: ■ **0**: The tool number is programmed ■ **1**: The tool name is programmed

Note

The control saves the tool usage file as a dependent file (***.dep**).

In the settings of the **Files** operating mode, you can specify whether the control displays dependent files in the file management.

Further information: Programming and Testing User's Manual

15.9 Preset table *.pr

Application

The **preset.pr** preset table allows you to manage presets. The active row in the preset table is used as a workpiece preset in the NC program and as the coordinate origin of the workpiece coordinate system **W-CS**.

Further information: "Presets in the machine", Page 148

Related topics

■ Setting and activating presets
  Further information: "Preset management", Page 179

Description of function

By default, the preset table has the name **preset.pr**, and is saved in the **TNC:\table** directory. The preset table is open in the **Tables** operating mode by default.

> ⚙ Refer to your machine manual.
>
> The machine manufacturer can define a different path for the preset table.
>
> In the optional machine parameter **basisTrans** (no. 123903), the machine manufacturer defines a specific preset table for each range of traverse.

Icons and buttons of the preset table

The preset table contains the following icons:

Icon	Meaning
⊕	Active row
🔒	Write-protected row

When you define a preset, the control opens a window with the following input options:

Button	Function
Set a preset	The control interprets the entered value as desired display value for the actual position. The control calculates the required table value from this.
	The entered value is active in the basic coordinate system **B-CS**.
	Further information: "Basic coordinate system B-CS", Page 175
	When you activate the edited preset, the control displays the entered value as actual position in the position display.
Correct	The control offsets the entered value against the actual table value. You can enter either a positive or a negative value.
	The entered value is active incrementally in the basic coordinate system **B-CS**.
Edit	The control accepts the entered value unchanged as table value.
	The entered value refers to the coordinate origin of the basic coordinate system **B-CS**.

Parameters of the preset table

The preset table contains the following parameters:

Parameter	Meaning
NO	Number of preset table row
DOC	Comment
X	X coordinate of preset
	Basic transformation relating to the basic coordinate system **B-CS** **Further information:** "Basic coordinate system B-CS", Page 175
Y	Y coordinate of preset
	Basic transformation relating to the basic coordinate system **B-CS** **Further information:** "Basic coordinate system B-CS", Page 175
Z	Z coordinate of preset
	Basic transformation relating to the basic coordinate system **B-CS** **Further information:** "Basic coordinate system B-CS", Page 175
ACTNO	Active workpiece preset
	The control automatically enters **1** in the active row.
LOCKED	Write-protection of the table row

> ⚙ Refer to your machine manual.
> In the optional machine parameter **CfgPresetSettings** (no. 204600), the machine manufacturer can block the setting of a preset in individual axes.

Write-protection for table rows

The **Lock record** button allows protecting any rows of the preset table against overwriting. The control enters the value **L** in the **LOCKED** column.

Further information: "Protecting table rows without a password", Page 297

Alternatively, the row can be protected with a password. The control enters the value **###** into the **LOCKED** column.

Further information: "Protecting table rows with a password", Page 297

The control displays an icon ahead of write-protected rows.

> ℹ If the control displays the value **OEM** in the **LOCKED** column, this column has been locked by the machine manufacturer.

NOTICE

Caution: Data may be lost!

Rows protected by a password can be unlocked by entering the selected password exclusively. Forgotten passwords cannot be reset. This would lock the protected rows permanently.

▶ Protecting table rows without a password is recommended
▶ Note down your passwords

15.9.1   Activating write protection

Protecting table rows without a password

To protect a table row without a password:

▶ Activate the **Edit** toggle switch

▶ Select the desired row

▶ Activate the **Lock record** toggle switch

> The control enters the value **L** in the **LOCKED** column.

> The control activates write-protection and displays an icon ahead of the row.

Protecting table rows with a password

NOTICE
Caution: Data may be lost!
Rows protected by a password can be unlocked by entering the selected password exclusively. Forgotten passwords cannot be reset. This would lock the protected rows permanently.
▶ Protecting table rows without a password is recommended
▶ Note down your passwords

To protect a table row with a password:

▶ Activate the **Edit** toggle switch

▶ Double-tap or double-click the **LOCKED** column of the desired row

▶ Enter the password

▶ Confirm your input

> The control enters the value **###** in the **LOCKED** column.

> The control activates write-protection and displays an icon ahead of the row.

15.9.2 Removing write protection

Unlocking table rows that are protected without a password

To unlock a table row that is protected without a password:

Edit
▶ Activate the **Edit** toggle switch

Lock record
▶ Deactivate the **Lock record** toggle switch
> The control removes the value **L** from the **LOCKED** column.
> The control deactivates the write protection and removes the icon ahead of the row.

Unlocking table rows that are protected with a password

NOTICE
Caution: Data may be lost!
Rows protected by a password can be unlocked by entering the selected password exclusively. Forgotten passwords cannot be reset. This would lock the protected rows permanently.
▶ Protecting table rows without a password is recommended
▶ Note down your passwords

To unlock a table row that is protected with a password:

Edit
▶ Activate the **Edit** toggle switch

▶ Double-tap or double-click the **LOCKED** column of the desired row
▶ Delete **###**
▶ Enter the password
▶ Confirm your input
> The control deactivates write-protection and removes the icon ahead of the row.

15.9.3 Creating a preset table in inches

If you define inches as the unit of measure in the **Machine Settings** menu item, the unit of measure of the preset table will not be adjusted automatically.

Further information: "The Machine Settings menu item", Page 333

To create a preset table in inches:

Shut down
- ▶ Restart the control
- ▶ Do not acknowledge **Power interrupted**
- ▶ Select the **Files** operating mode

- ▶ Open the **TNC:\table** folder
- ▶ Rename the original file **preset.pr** (e.g., as **preset_mm.pr**)
- ▶ Select the **Tables** operating mode

- ▶ Select **Create new table**
- > The control opens the **Create new table** window.
- ▶ Select the **pr** folder
- ▶ Select INCH as the unit of measure if necessary
- ▶ Select the desired prototype

- ▶ Select a path
- > The control opens the **Save as** window.
- ▶ Select the **table** folder
- ▶ Enter the name **preset.pr**

Create
- ▶ Select **Create** twice
- > The control opens the **Presets** tab in **Tables** operating mode.

Shut down
- ▶ Restart the control

CE
- ▶ Acknowledge **Power interrupted** with the **CE** key

- ▶ Select the **Presets** tab in **Tables** operating mode
- > The control uses the newly created table as a preset table.
- > The control shows INCH as the unit of measure in the dialog bar of the workspaces.

Notes

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. If the previous value is kept, there is a danger of collision!

▶ Before activating a preset, check whether all columns contain values.

▶ For undefined columns, enter values (e.g., **0**)

▶ As an alternative, have the machine manufacturer define **0** as the default value for the columns

---

- To optimize the file size and the processing speed, keep the preset table as short as possible.
- New rows can be inserted only at the end of the preset table.
- If you edit the value of the **DOC** column, then the preset must be reactivated. Only then does the control apply the new value.
  Further information: "Activating presets", Page 180
- If a manual probing process or an NC program is interrupted or stopped, you cannot edit the preset table. When you double-tap or double-click a table cell the control shows the **Editing not possible. Perform internal stop?** window. If you select **Yes**, the control may lose touch points or modally active program information.

Notes about machine parameters

- In the optional machine parameter **initial** (no. 105603), the machine manufacturer defines a default value for every column of a new row.
- If the unit of measure of the preset table does not match the unit of measure defined in the machine parameter **unitOfMeasure** (no. 101101), the control displays a message in the dialog bar of the **Tables** operating mode.

16

Electronic handwheel

16.1 Fundamentals

Application

The electronic handwheel allows you to traverse the axes without needing to stand at the operating panel. You can also use the handwheel to perform control functions, such as setting up the machine or influencing program run.

⚙	Refer to your machine manual. The machine manufacturer defines which functions are available on the handwheel and how the control evaluates these functions. This chapter describes the standard features of the handwheel.

Related topics

- Incremental jog positioning

  Further information: "Incremental jog positioning of axes", Page 129
- Touch probe functions in the **Manual** operating mode (#17 / #1-05-1)

  Further information: "Touch probe functions in the Manual operating mode (#17 / #1-05-1)", Page 209

Overview

The control supports the following handwheels:

Handwheel	Meaning	Further information
HR 130, HR 180	Handwheel mounted on the operating panel	
HR 510, HR 510 FS	Handwheel without display	Page 303
HR 520, HR 520 FS HR 550 FS	Handwheel with display with wireless transmission	Page 306

Notes

> **⚠ DANGER**
>
> **Caution: hazard to the user!**
>
> 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 machine manufacturer defines which axes you can move with the handwheel. Your machine manufacturer can also place the virtual axis **VT** on an axis key.
- If the handwheel is active, the control shows an icon for the selected axis in the **Positions** workspace. The icon indicates whether you can move the axis with the handwheel.

  Further information: "The Positions workspace", Page 95

16.2   Handwheel without display

Application

This chapter contains supplementary information about the HR 510 and HR 510 FS handwheels without display.

Related topics

- Overview of the available handwheels
  Further information: "Overview", Page 302
- Handwheels with display
  Further information: "Handwheel with display", Page 306

Description of function

Operating elements

A handwheel without display features the following operating elements:

1   Axis keys
2   The **actual position capture** key
    Further information: "Creating an NC block with the current position", Page 305
3   Keys for the speed level
    Further information: "Speed levels", Page 304
4   Traverse direction keys
5   Keys which the machine manufacturer can assign, such as Spindle ON, **NC Start** or **NC stop**
6   Handwheel permissive buttons
    Further information: "Activating and deactivating a handwheel", Page 304
7   Dial
8   **Emergency stop** key

Speed levels

The handwheel offers three keys with pre-defined speed levels that you can choose from.

The speed level influences the following values:

- Distance that the control moves an axis by when you rotate the wheel by one detent stop
- Feed rate at which the control moves the axis when you press an axis-direction key

> **i** You use a key to specify both the feed rate as well as the speed level for the distance moved. However, the control uses different, independent values, depending on whether you rotate the wheel or press a key.

Key	Meaning
⌇⌇⌇	Low speed level
	Example: Distance of 0.001° or 0.001 mm/inch
⌇⌇	Medium speed level
	Example: Distance of 0.01° or 0.01 mm/inch
∿	High speed level
	Example: Distance of 0.1° or 0.1 mm/inch

> **⚙** Refer to your machine manual.
> The machine manufacturer defines the values of the speed levels for each axis.

16.2.1 Activating and deactivating a handwheel

To activate a handwheel without display:

⌬
- ▶ Press the **Handwheel** key on the control
- > The NC control activates the handwheel and changes the symbol in the **Manual** operating mode.
- > The control displays a handwheel symbol next to the currently selected axis in the **Position** workspace.

To deactivate a handwheel without display:

⌬
- ▶ Press the **Handwheel** key on the control

> **i** As an alternative to the **Handwheel** key, you can also use the **Handwheel** toggle switch in the function bar of the control.

16.2.2 Creating an NC block with the current position

⚙ Refer to your machine manual.
The machine manufacturer can assign any keys on the handwheel. Your handwheel therefore might not have an **actual position capture** key.

To create an NC block using the handwheel:

🖐 ▶ Select the **Manual** operating mode

🖐▶ ▶ Select the **MDI** application
▶ If necessary, select the NC block after which you want to insert the NC block
▶ Activate the handwheel

⊞ ▶ Press the **actual position capture** key
> The control inserts the actual position of the active axis.

16.2.3 Incremental jog positioning using handwheels without display

Incremental jog positioning allows you to move the selected axis by a defined value each time you press a key. In order to perform incremental jog positioning using a handwheel without display, you must first define the jog increment on the control.

ℹ Incremental jog positioning works only for traverse movements using the axis-direction keys.

To perform incremental jog positioning using a handwheel without display:

🖐 ▶ Select the **Manual** operating mode

🖐 ▶ Select the **Manual operation** application

Jog increment ▶ Select **Jog increment**
> The control opens the **Positions** workspace, if necessary, and shows the **Jog increment** area.
▶ Enter the jog increment for linear axes

⊛ ▶ Press the **Handwheel** key on the control
> The control activates the handwheel.

X ▶ Press an axis key

+ ▶ Press a traverse direction key
> The control moves the axis by the defined jog increment.

16.3 Handwheel with display

Application

This chapter contains information specific to the HR 520, HR 520 FS and HR 550 FS handwheels with display. Handwheels with display can perform more functions than handwheels without display.

Related topics

- Overview of handwheels
  Further information: "Overview", Page 302
- Handwheels without display
  Further information: "Handwheel without display", Page 303

Description of function

Operating elements
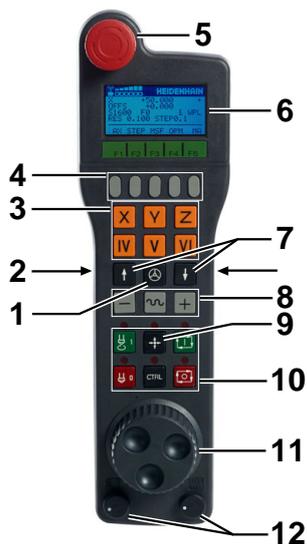
A handwheel with display provides the following operating elements:

1 Handwheel activation key

   Further information: "Activating and deactivating a handwheel with display", Page 311

2 Handwheel permissive buttons on the sides

3 Axis keys

4 Soft-key selection keys

   Further information: "Handwheel soft keys", Page 308

5 **Emergency stop** key

6 Display

   Further information: "Contents of display", Page 307

7 Speed level

   Further information: "Speed levels", Page 309

8 Keys for traverse direction and rapid traverse

9 The **actual position capture** key

   Further information: "Creating an NC block with the current position", Page 305

10 Keys which the machine manufacturer can assign, such as Spindle ON, **NC Start** or **NC stop**

11 Dial

12 Potentiometers for spindle speed and feed rate

Contents of display
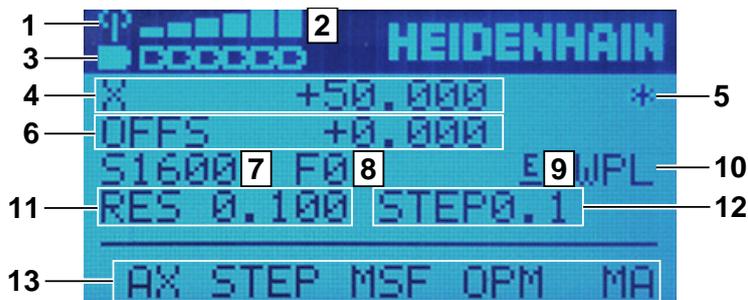
The display of a handwheel consists of the following areas:

1   Handwheel is in the docking station or radio mode is active
    Only with HR 550 FS wireless handwheel

2   Field strength, up to six bars
    Only with HR 550 FS wireless handwheel

3   Charge status of the rechargeable battery, up to six battery symbols
    Only with HR 550 FS wireless handwheel

4   Selected axis and current position

5   Control-in-operation
    Program run has been started or axis is in motion

6   Without function for TNC7 go

7   Current speed of the active spindle

8   Current feed rate of the selected axis
    During program run: current contouring feed rate

9   Pending error message

10 Without function for TNC7 go

11 Traverse per wheel detent stop
    Further information: "Speed levels", Page 309

12 Incremental jog active or inactive, and jog increment
    Further information: "Incremental jog positioning", Page 313

13 Handwheel soft keys
    Further information: "Handwheel soft keys", Page 308

Handwheel soft keys

You can use the handwheel soft keys to select the following functions:

Soft key	Key	Meaning
AX	F1	Select machine axis
STEP	F2	Active or deactivate incremental jog positioning, and select the jog increment **Further information:** "Incremental jog positioning", Page 313
MSF	F3	Define cutting data, miscellaneous functions, and presets **Further information:** "Defining the spindle speed S", Page 312
OPM	F4	Select operating mode
MA	F5	Execute machine-specific functions (such as switching magazine pockets)
MOP	F3	Select manual options Only if program run was interrupted by an **NC stop**

Operating modes

Press **OPM** to choose between the following operating modes:

Soft key	Key	Meaning
MAN	F1	**Manual** operating mode
MDI	F2	**MDI** application in the **Manual** operating mode
RUN	F3	**Program Run** operating mode
SGL	F4	**Single Block** mode in the **Program Run** operating mode

Functions during program run

Press **MOP** to choose between the following functions:

Soft key	Key	Meaning
MAN	F1	Manual traverse
STOP	F4	Internal stop

Press **MAN** at this level to choose between the following functions:

Soft key	Key	Meaning
REPO	F1	Return to the contour
3D	F2	Settings of the Tilt working plane function

Speed levels

You use the up and down arrow keys to choose the speed level.

The handwheel shows the value of the selected level in the display after **RES**. If you rotate the wheel by one detent stop, the control moves the axis by the value in the unit of measure for that axis.

Key	Meaning
↑	Increase speed level
↓	Decrease speed level

ⓘ On handwheels with display the speed level defines only the distance per detent stop; it does not define the feed rate for traverse movements with axis-direction keys.

You use the **MSF** soft key to define the feed rate.

Further information: "Defining the handwheel feed rate F", Page 312

Special features of the HR 550 FS wireless handwheel

With the HR 550 FS wireless handwheel you can move farther away from the machine operating panel than with cable-connected handwheels. The HR 550 FS wireless handwheel thus provides an important benefit, in particular for large machines.

The HRA 551 FS handwheel holder and the HR 550 FS handwheel together form one functional unit.

HR 550 FS handwheel   HRA 551 FS handwheel holder

The HR 550 FS wireless handwheel features a rechargeable battery. The battery starts charging when you place the handwheel into the holder.

The HR 550 FS can be operated by battery for up to eight hours before it needs recharging. A completely discharged handwheel takes approx. three hours for a full charge. When you aren't using the HR 550 FS, always place it in the handwheel holder. This charges the handwheel battery constantly and a direct connection with the emergency-stop circuit is provided.

When the handwheel is in its holder, it provides the same functionality as during radio mode. This allows you to use a completely discharged handwheel.

> Always dock an HR 550 FS in its HRA 551 FS when not in use. This has the following benefits:
> - The safety self-test can be performed:
> - The NiMH rechargeable battery of the handwheel is always fully charged.
> - An unexpected emergency stop reaction due to an empty NiMH rechargeable battery is avoided.
> - Confusion between different wireless handwheels is avoided.

> Clean the contacts of the handwheel holder and handwheel regularly to ensure their proper functioning.
>
> You can configure wireless handwheels in the **TNCdiag** application.
>
> **Further information:** "Setting up a wireless handwheel ", Page 315
>
> If the control has triggered an emergency stop, you must reactivate the handwheel.

If you happen to get close to the limit of the transmission range, the HR 550 FS will set off a vibrating alarm. If this occurs, you must reduce the distance to the handwheel holder.

16.3.1 Activating and deactivating a handwheel with display

> ### NOTICE
>
> **Caution: Possible damage to the workpiece!**
>
> When toggling between the machine operating panel and the handwheel, the feed rate may be reduced. This can cause visible marks on the workpiece.
>
> ▶ Retract the tool before switching
> ▶ Before switching, check whether the settings of the feed-rate potentiometers match

To activate a handwheel with display:

▶ Press the **Handwheel** key on the handwheel
> The NC control activates the handwheel and changes the symbol in the **Manual** operating mode.
> The control displays a handwheel symbol next to the currently selected axis in the **Position** workspace.
▶ If necessary, adjust the setting of the feed-rate potentiometer

To deactivate a handwheel with display:

▶ Press the **Handwheel** key on the handwheel

> The control also activates or deactivates the feed-rate potentiometer.
>
> If the feed rate before switching is higher than the feed rate after switching, the control automatically reduces the feed rate to the smaller value.
>
> If the feed rate before switching is less than the feed rate after switching, the control automatically freezes the feed rate. In this case, you must turn the feed-rate potentiometer back to the previous value because only then will the activated feed-rate potentiometer take effect.

16.3.2 Creating an NC block with the current position

> Refer to your machine manual.
> The machine manufacturer can assign any keys on the handwheel. Your handwheel therefore might not have an **actual position capture** key.

To create an NC block using the handwheel:

▶ Select the **Manual** operating mode

▶ Select the **MDI** application
▶ If necessary, select the NC block after which you want to insert the NC block
▶ Activate the handwheel

▶ Press the **actual position capture** key
> The control inserts the actual position of the active axis.

16.3.3 Defining the spindle speed S

To define the spindle speed **S** of the active spindle when using a handwheel with display:

▶ Press the **MSF** soft key on the handwheel
▶ Press the **S** soft key on the handwheel
▶ Select the desired spindle speed by pressing the **F1** or **F2** keys
> The handwheel shows the defined spindle speed in the display after **S**.
▶ Press the **NC Start** key
> The control activates the defined spindle speed.

> **i** If you press and hold the **F1** or **F2** key, the handwheel counts the value up or down. The longer you press the key, the greater the counting step.
>
> If you additionally press the **CTRL** key, the handwheel starts with a larger counting step.

16.3.4 Defining the handwheel feed rate F

To define the feed rate **F** when using a handwheel with display:

▶ Press the **MSF** soft key on the handwheel
▶ Press the **F** soft key on the handwheel
▶ Select the desired feed rate by pressing the **F1** or **F2** keys
▶ Press **OK**
▶ Confirm the new feed rate by pressing the **OK** soft key on the handwheel

[X] ▶ Press an axis key

[+] ▶ Press a traverse direction key
> The control moves the axis at the defined feed rate.

> **i** If you press and hold the **F1** or **F2** key, the control will increase the counting increment by a factor of 10 each time it reaches a value divisible by 10.
>
> By additionally pressing the **CTRL** key, you can increase the counting increment by a factor of 100 when pressing **F1** or **F2**.

16.3.5 Incremental jog positioning

Incremental jog positioning allows you to move the selected axis by a defined value each time you press a key.

> **i**   ■ Incremental jog positioning works only for traverse movements using the axis-direction keys.
>   ■ The control compares the settings for incremental jog positioning defined for the handwheel and the control.

To perform incremental jog positioning using a handwheel with display:

▶ Press the **STEP** soft key on the handwheel

▶ Press the **ON** soft key on the handwheel

> The control activates incremental jog positioning.

▶ Select the desired jog increment by pressing the **F1** or **F2** keys

> The handwheel shows the defined jog increment in the display after **STEP**.

> **i**   The smallest possible increment is 0.0001 mm (0.00001 inches). The largest possible increment is 10 mm (0.3937 inches).

▶ Confirm the jog increment by pressing the **OK** soft key on the handwheel

| X | ▶ Press an axis key |

| + | ▶ Press a traverse direction key |
| | > The control moves the axis by the defined jog increment. |

> **i**   If you press and hold the **F1** or **F2** key, the control will increase the counting increment by a factor of 10 each time it reaches a value divisible by 10.
>   By additionally pressing the **CTRL** key, you can increase the counting increment by a factor of 100 when pressing **F1** or **F2**.

Notes on wireless handwheels

> ## ⚠ DANGER
>
> **Caution: hazard to the user!**
>
> 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 handwheel and the handwheel holder after an operating time of 120 hours at the latest so that the control can run a functional test the next time it is restarted (only for handwheels 598515-03, 606622-03 and holder 731928-02)
> ▶ 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)

> ## *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

■ The control displays a warning if you connect a wireless handwheel with an already selected radio channel.

16.4 Setting up a wireless handwheel

Application

You can configure the HR 550 FS wireless handwheel in the **Setup for wireless handwheel** application.

Related topics

- Electronic handwheel

  Further information: "Electronic handwheel", Page 301
- HR 550 FS wireless handwheel

  Further information: "Special features of the HR 550 FS wireless handwheel", Page 310
- TNCdiag

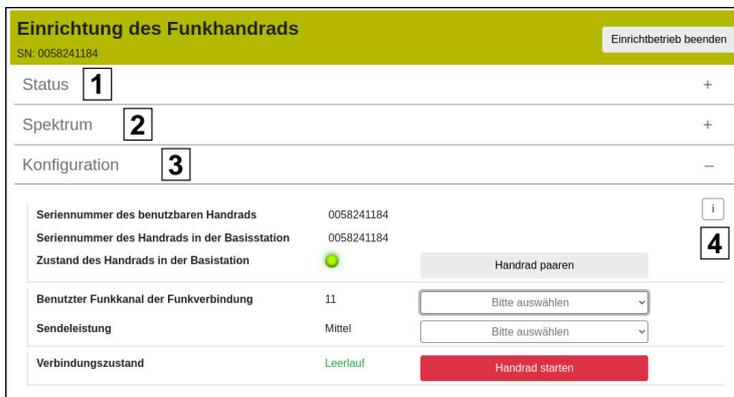 **Further information:** "TNCdiag", Page 388

Requirements

- Machine with a handwheel holder

  The machine manufacturer installs the handwheel holder on the machine.

Description of function

To navigate to this function:

Home ▶ Settings ▶ Machine Settings ▶ Set Up Wireless Handwheel

TNCdiag within the **Setup for wireless handwheel** application

TNCdiag shows the following areas:

1 **Status**

Information about the transmission quality

If the reception quality of the wireless connection is poor, the control triggers an emergency stop. Safe stopping of the axes cannot be ensured when the reception quality is poor.

2 **Spectrum**

Frequency of the individual radio channels

The radio channel with the shortest column has the least amount of radio traffic. The radio channel recommended for the wireless handwheel is marked with a green circle.

3 **Configuration**

■ **Pair handwheel**

Assign the wireless handwheel to the handwheel holder

■ **Channel used for radio connection**

In the selection menu, choose **Best channel** to select the recommend radio channel for the wireless handwheel

■ **Transmitter power**

Select Transmitter power in the selection menu. The lower the transmission power, the smaller the range of the wireless handwheel.

■ **Connection status**

Once **TNCdiag** continuously shows **Active** as the connection status, configuration is complete.

4 **Information**

Each area shows the **Information** icon. When you select the icon, **TNCdiag** displays the description of the settings.

16.4.1 Setting up a new wireless handwheel

To set up a new wireless handwheel:

▶ Place the handwheel in the handwheel holder

⌂ ▶ Select the **Home** operating mode

⚙ ▶ Select the **Settings** application

▶ Select **Machine Settings**

▶ Double-tap or double-click **Set Up Wireless Handwheel**
> The control opens the **Setup for wireless handwheel** application in **TNCdiag**.
▶ Select **Pair handwheel**
> **TNCdiag** briefly shows **Active** under **Connection status**.
▶ In the **Channel used for radio connection** selection menu, choose **Best channel** for the channel
▶ Select the transmitter power in the **Transmitter power** selection menu (e.g., **Medium**)
▶ Select **Start handwheel**
> **TNCdiag** activates the handwheel.
> **TNCdiag** dims the **Channel used for radio connection** and **Transmitter power** selection menus.

ⓘ If an error occurs during configuration, **TNCdiag** colors the **Configuration** area red.

In order to read the error details, switch to an operating mode, such as the **Home** operating mode.

17

Extended
Workspace

17.1   Extended Workspace

Application

With Extended Workspace you can use an additional attached monitor as a second screen of the control. That way you can use the additional monitor independently of the control's user interface and also to show the control's applications.

Related topics

■ ITC hardware expansion

Further information: "Hardware enhancements", Page 56

Requirement

■ Additional attached monitor configured by the machine manufacturer as Extended Workspace

Refer to your machine manual.

Description of function

Here are some functions you can perform with Extended Workspace:

■ Opening files from the control (e.g., drawings)
■ Opening windows from HEROS functions in addition to the control's user interface

Further information: "HEROS menu", Page 431

18

**Integrated
functional safety
(FS)**

Application

The safety design of integrated functional safety (FS) for machines with HEIDENHAIN controls offers supplementary software safety functions in addition to the mechanical safety features of the machine. For example, the integrated safety design automatically reduces the feed rate when you perform operations with open guard doors. The machine manufacturer can modify or expand the FS safety design.

Requirements

- On controls with **SIK**:
  - Software option 160 (Integrated FS: Basic) or software option 161 (Integrated FS: Full)
  - Software options 162 to 166 (Add. FS Ctrl. Loop) or software option 169 (Add. FS Full) as needed

    Whether you need these software options depends on the machine's number of motors.
- On controls with **SIK2**:
  - Integrated FS: Basic software option (#6-30-1)
  - Integrated FS: Full software option as needed (#6-30-2*)

    If your control is equipped with **SIK2**, software option #6-30-1 will enable four safe axes. You can order software option #6-30-2* multiple times and thus enable up to six additional safe axes.
- The machine manufacturer must adapt the FS safety design to the machine.

Description of function

Every machine tool user is exposed to certain risks. While protective devices can prevent access to dangerous locations, the user must also be able to work on the machine without this protection (e.g., guard door opened).

Safety functions

To ensure that the requirements for operator protection are met, integrated functional safety (FS) provides standardized safety functions. The machine manufacturer uses the standardized safety functions for implementing functional safety (FS) for the machine in question.

You can track the active safety functions in the axis status of functional safety (FS).

Further information: "The Axis status menu item", Page 323

Description	Meaning	Short description
SS0, SS1, SS1D, SS1F, SS2	Safe Stop	Safe stopping of motors using different methods
STO	Safe Torque Off	The power supply to the motor is interrupted. Provides protection against unexpected start of the motors
SOS	Safe Operating Stop	Safe operating stop. Provides protection against unexpected start of the motors
SLS	Safely Limited Speed	Safely limited speed. Prevents the motors from exceeding the specified speed limits when the guard door is opened
SLP	Safely Limited Position	Safely limited position. Monitors safe axes to keep them within the limit values of a defined area
SBC	Safe Brake Control	Dual-channel control of the motor holding brakes

Safety-related operating modes of functional safety (FS)

Functional safety (FS) of a control offers various safety-related operating modes. The safety-related operating mode with the lowest number has the highest safety level.

Depending on how the machine manufacturer implements them, the following safety-related operating modes are available:

> ⚙ Refer to your machine manual.
> The machine manufacturer must adapt the safety-related operating modes to each machine.

Icon	Safety-related operating mode	Short description
SOM 1	Operating mode **SOM_1**	Safe operating mode 1: Automatic mode, production mode
SOM 2	Operating mode **SOM_2**	Safe operating mode 2: Setup mode
SOM 3	Operating mode **SOM_3**	Safe operating mode 3: Manual intervention; only for qualified users
SOM 4	Operating mode **SOM_4** This function must be enabled and adapted by the machine manufacturer.	Safe operating mode 4: Advanced manual intervention, process monitoring, only for qualified users

Functional safety (FS) in the Positions workspace

On a control with functional safety (FS), the monitored operating states of the speed **S** and feed rate **F** are displayed in the **Positions** workspace. If a safety function is triggered while in a monitored state, the control stops the feed movement and the spindle or reduces the speed (e.g., if a guard door is opened).

Further information: "Axis display and position display", Page 96

The Functional safety application

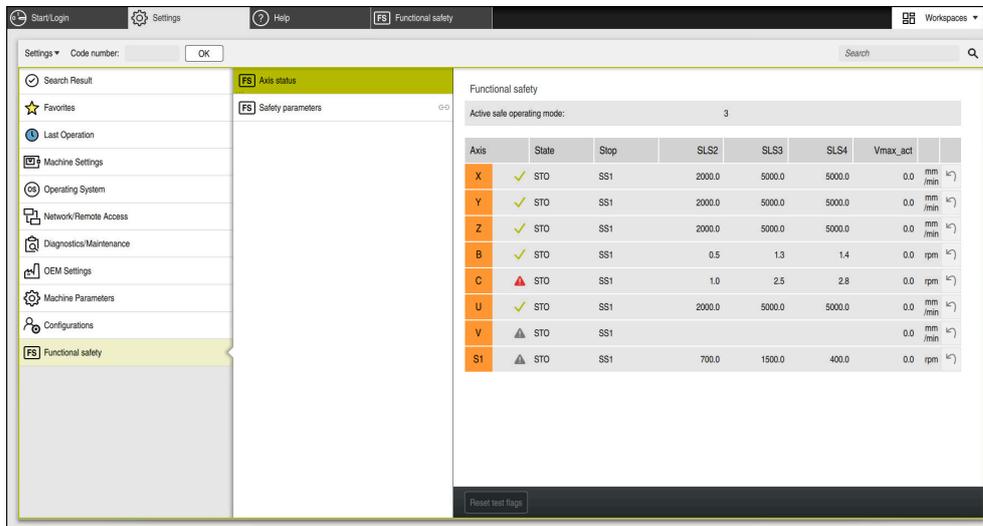> ⚙ Refer to your machine manual.
>
> The machine manufacturer configures the safety functions in this application.

In the **Functional safety** application in the **Home** operating mode, the control provides information about the status of the individual safety functions. In this application you can see whether individual safety functions are active and have been accepted by the control.

The **Overview** workspace in the **Functional safety** application

The **Axis status** menu item

The **Axis status** menu item in the **Settings** application

In the **Axis status** menu item of the **Settings** application, the control provides the following information about the status of the individual axes:

Field	Meaning
Axis	Configured axes of the machine
State	Active safety function
Stop	Stop reaction **Further information:** "Functional safety (FS) in the Positions workspace", Page 322
SLS2	Maximum speed or feed-rate values for **SLS** in the **SOM_2** operating mode
SLS3	Maximum speed or feed-rate values for **SLS** in the **SOM_3** operating mode
SLS4	Maximum speed or feed-rate values for **SLS** in the **SOM_4** operating mode This function must be enabled and adapted by the machine manufacturer.
Vmax_act	Currently valid speed or feed-rate limit These are either values from the **SLS** settings or from the SPLC If values are greater than 999 999, the control displays **MAX**.

The control shows the following icons and buttons:

Icon or button	Meaning
↰	Reset the test status of the selected axis
Reset test flags	Reset the test statuses of all axes

---

i   ■ Resetting of the test status is a function for the Service department. Use this function only if instructed by HEIDENHAIN or the machine manufacturer.

■ To reset the test status of axes, you need the NC.ApproveFsAxis right. This right is only available if user administration is active.

Further information: "User administration", Page 401

Further information: "User administration roles and rights", Page 455

Test status of the axes

In order for the control to ensure safe operation of the axes, it checks all monitored axes when the machine is switched on.

The control checks whether the position of an axis matches the position directly after shutdown. If a deviation is detected, the control marks the respective axis in the position display with a red warning triangle.

If checking of individual axes fails when starting the machine, you can check the axes manually.

Further information: "Checking axis positions manually", Page 325

The control indicates the test status of the individual axes with the following icons:

Icon	Meaning
✓	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 axis positions manually", Page 325
⚠	The axis is not monitored by functional safety (FS) or is not configured as a safe axis. The axis is monitored by functional safety (FS), but the **SLP** safety function is deactivated. In machine parameter **safeAbsPosition** (no. 403130), the machine manufacturer defines whether the **SLP** safety function is activated for an axis.

18.1 Checking axis positions manually

⚙ Refer to your machine manual.
This function must be adapted by your machine manufacturer.
The machine manufacturer defines the test position.

To check the position of an axis:

▶ Select the **Manual** operating mode

▶ Select the **Move to ref. point** application

▶ Select **Axis position** in the **Referencing** workspace
▶ Select the desired axis
▶ Press the **NC start** key

> The axis moves to the test position.
> After the test position has been reached, the control issues a message.
▶ Press the **permissive button** on the machine operating panel
> The control displays the axis as a tested axis.

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

ℹ In the **Referencing** workspace, you can switch as desired between the **Referencing** and **Axis position** modes.

Notes

- Machine tools with HEIDENHAIN controls may be equipped with integrated functional safety (FS) or with external safety. This chapter refers exclusively to machines with integrated functional safety (FS).
- The control performs repeated self-tests to detect defective cables, for example. The machine manufacturer defines at what intervals the control will perform the self-tests. When a self-test of the control is active, the control displays an icon in the information bar. The control cannot perform any axis movements while a self-test is active.

19

The Settings application

19.1 Overview

The **Settings** application includes the following groups with menu items:

Icon	Category	Icon	Menu item
⊡	**Machine Settings**	⊡	**Machine Settings** **Further information:** "The Machine Settings menu item", Page 333
		ⓘ	**General Information** **Further information:** "The General Information menu item", Page 336
		▭	**SIK** **Further information:** "The SIK menu item", Page 337
		🕐	**Machine Times** **Further information:** "The Machine Times menu item", Page 340
		⚒	**Overview of touch probes** **Further information:** "Overview of touch probes menu item", Page 341
		Ⓥ	**Adjustment of analog voltage offset** **Further information:** "The Adjustment of analog voltage offset menu item", Page 344
		📶	**Set Up Wireless Handwheel** **Further information:** "Setting up a wireless handwheel ", Page 315
OS	**Operating System**	📅	**Date/Time** **Further information:** "The Adjust system time window", Page 345
		A文	**Language/Keyboards** **Further information:** "Conversational language of the control", Page 346
		OSⓘ	**About HeROS** **Further information:** "Information on licensing and use", Page 51
		🔒	**SELinux** **Further information:** "SELinux security software", Page 348
		👥	**UserAdmin** **Further information:** "The User administration window", Page 410
		👤	**Current User** **Further information:** "The Active user window", Page 411
		👆	**Touchscreen Configuration** You can select the touchscreen sensitivity and define whether touch points should be shown or hidden.

Icon	Category	Icon	Menu item
⛁	**Network/Remote Access**	⤙	**Shares** **Further information:** "Network drives on the control", Page 349
		⛁	**Network** **Further information:** "Ethernet interface", Page 353
		▤	**PKI Admin** Manage certificates for the control (e.g., for **OPC UA NC Server**) **Further information:** "PKI Admin", Page 360
		⇆	**OPC UA** **Further information:** "OPC UA NC Server (#56-61 / #3-02-1*)", Page 362
		⇆	**DNC** **Further information:** "The DNC menu item", Page 370
		🖶	**Printer** **Further information:** "Printers", Page 373
		vnc	**VNC** **Further information:** "The VNC menu item", Page 376
		vnc	**Real VNC Viewer** Connect to a remote device via a VNC server Available only to network specialists
		⛉	**Firewall** **Further information:** "Firewall", Page 379

Icon	Category	Icon	Menu item
⬚	Diagnostics/Maintenance	>_	**Terminal program** Enter and execute console commands
		⊟	**HeLogging** Define settings for internal diagnostic files
		🔍	**Portscan** **Further information:** "Portscan", Page 384
		⌲	**perf2** Check processor load and process load
		🗗	**TNCdiag** **Further information:** "TNCdiag", Page 388
		∿	**TNCscope** Available only to authorized specialists
		🗄	**NC/PLC Backup** **Further information:** "Backup and restore", Page 385
		🗄	**NC/PLC Restore** **Further information:** "Backup and restore", Page 385
		✦	**Touchscreen Cleaning** The control disables the touchscreen for input for 90 seconds.
		↻	**Update the documentation** **Further information:** "Update the documentation", Page 389
		⬚	**Expanded logging** If this function is active, the control will save graphics journal data. These data may be needed by the HEIDENHAIN Customer Service for diagnostic purposes. If you activate this function, you need to restart the control. Deactivate this function after the fault has been fixed to reduce the amount of memory used.
🏭	OEM Settings		Settings for the machine manufacturer
⚙	Machine Parameters		The group contains machine parameters that can be edited, depending on your rights (e.g., **MPs for setters**). **Further information:** "Machine parameters", Page 391
👤	Configurations	👤	**Configurations** **Further information:** "Configuring the control's user interface", Page 398

Icon	Category	Icon	Menu item
[FS]	**Functional safety**	[FS]	**Axis status** **Further information:** "The Axis status menu item", Page 323
		[FS]	**Safety parameters** **Further information:** "The Functional safety application", Page 322

19.2 Code numbers

Application

The top part of the **Settings** application contains the **Code number:** input field. This input field is accessible from every group.

Description of function

You can enable the following functions or areas with code numbers:

Code number	Meaning
123	Editing machine-specific user parameters **Further information:** "Machine parameters", Page 391
🏠	When this code number is active, the control displays a different icon for the **Home** operating mode.
555343	Special functions for programming with variables **Further information:** Programming and Testing User's Manual Special functions defining the machine behavior **Further information:** Programming and Testing User's Manual
0	Resetting active code numbers

> ⓘ The control indicates whether the caps lock key is pressed during entry. This helps to avoid incorrect entries.

19.3 The Machine Settings menu item

Application

In the **Machine Settings** menu item of the **Settings** application, you can define the settings for simulation and program run.

Related topics

- Graphic settings for simulation

  Further information: Programming and Testing User's Manual

Description of function

To navigate to this function:

Settings ▶ **Machine Settings** ▶ **Machine Settings**

The Unit of Measure area

In the **Unit of Measure** area you can choose between mm and inch.

- Metric system: e.g. X = 15.789 (mm), the value is displayed to 3 decimal places
- Inch system: e.g. X = 0.6216 (inches), the value is displayed to 4 decimal places

If the display in inches is active, the control also displays the feed rate in inches/min. In an inch-based program, you must multiply the feed rate by 10 before entering it.

The Machine and Simulation areas

Channel Settings

You can define the following settings:

Setting	Meaning
Active Kinematics	Use the **Active Kinematics** function to change the kinematics model for the machine and the simulation. This way you can test NC programs that, for example, have been programmed for other machines.
	The control offers a selection menu with all available kinematics models. The machine manufacturer defines which kinematics models you can choose.
	The control displays the active kinematics model in the **Machine** mode of the **Simulation** workspace.
Generate tool-usage file	The control uses the tool-usage file to check tool usage.
	Further information: "Tool usage test", Page 169
	You select when the control should generate a tool-usage file:
	■ **Never**
	The control does not generate a tool-usage file.
	■ **Once**
	The next time you simulate or run an NC program, the control will generate a tool-usage file once.
	■ **Always**
	When you simulate or run an NC program, the control will generate a tool-usage file each time.

Oper. station for program interruption

The control displays this function only in the **Machine** area.

> ⚙ Refer to your machine manual.
>
> The machine manufacturer uses the optional machine parameter **userControl** (no. 144101) to define if this area is available.

You can define the following settings for the **Manual traverse** during program run:

Setting	Meaning
Standard operating station	You select the operating element to be used for moving the axes: ■ **MB** machine operating panel ■ **HR** handwheel
Remember last operating station	If the toggle switch is active, the control will not consider the selected **Standard operating station**. The control remembers the state of the **Handwheel** toggle switch from the last manual traverse, and activates the associated operating element.

Further information: "Manual traverse during an interruption", Page 239

Traverse Limits

The control displays this function only in the **Machine** area.

Use the **Traverse Limits** function to limit the possible traverse path of an axis. You can define traverse limits for each axis (e.g., to protect an indexing head from collision).

The **Traverse Limits** function consists of a table with the following contents:

Column	Meaning
Axis	The TNC displays each axis of the active kinematics model in a row.
Status	If you have defined one or both limits, the control displays the contents **Valid** or **Invalid**.
Lower Limit	You define the lower traverse limit of the axis in this column. You can enter up to four decimal places.
Upper Limit	You define the upper traverse limit of the axis in this column. You can enter up to four decimal places.

The defined traverse limits are valid across power cycles of the control, until you delete all values from the table.

The following general conditions apply to the traverse limit values:

■ The lower limit must be smaller than the upper limit.

■ The upper and lower limit may not both equal 0.

Notes

NOTICE
Danger of collision!
You can also select any stored kinematics model as the active machine kinematics. The control then executes all manual movements and machining operations using the selected kinematics. All subsequent axis movements pose a risk of collision!
▶ Use the **Active Kinematics** function for the simulation only
▶ Use the **Active Kinematics** function for selecting the active machine kinematics only if required

- In the optional machine parameter **enableSelection** (no. 205601), the machine manufacturer defines for each kinematics model whether the **Active Kinematics** function can be selected.

- Use the **+**, **-**, *****, **/**, **(**, and **)** keys for calculations in the numerical input fields.

- You can open the tool-usage file in the **Tables** operating mode.
  Further information: "Tool usage file", Page 293

19.4 The General Information menu item

Application

In the **General Information** menu item of the **Settings** application, the control provides information about the control and the machine. If, for example, the HEIDENHAIN service department requires the NC software number, you will find that number in this area.

Description of function

To navigate to this function:

Settings ▶ Machine Settings ▶ General Information

The Version Information area

The control displays the following information:

Sub-area	Meaning
HEIDENHAIN	■ **Control Model** Designation of the control ■ **NC-SW** Number of the NC software ■ **NCK** Version of the software Is managed by HEIDENHAIN
PLC	**PLC-SW** Number or name of the PLC software Is managed by the machine manufacturer
Operating System	■ **HEROS version** ■ **Yocto version** Is managed by HEIDENHAIN

The machine manufacturer can add further software numbers .

If the machine manufacturer has defined a version number for at least one OEM cycle, the control displays this information in the **OEM cycles** sub-area.

The Info about machine manufacturer area

The control shows the contents of the optional machine parameter **CfgOemInfo** (no. 131700). The control displays this area only if the machine manufacturer defines this machine parameter.

Further information: "Machine parameters in conjunction with OPC UA", Page 363

The Machine information area

Sub-area	Meaning
Hardware	**RAM** RAM of the main computer

The control also shows the contents of the optional machine parameter **CfgMachineInfo** (no. 131600). The control displays this area only if the machine operator defines this machine parameter.

Further information: "Machine parameters in conjunction with OPC UA", Page 363

19.5 The SIK menu item

Application

Use the **SIK** menu item of the **Settings** application to view control-specific information (e.g., the serial number and the available software options).

Related topics

- Software options on the control
  Further information: "Software options", Page 49

Description of function

To navigate to this function:

Settings ▶ Machine Settings ▶ SIK

The **SIK** menu item consists of the following areas:

Area	Contents
SIK Information	The control displays the following information: ■ **Serial Number** ■ **ID number** ■ **Control Model** ■ **Performance Class** ■ **Features** ■ **Status** The control shows whether it is equipped with a **SIK** or **SIK2**. ■ **Temporarily enable options / Disable options**
Machine manufacturer key	The machine manufacturer can define a manufacturer-specific password for the control.
General key Only with **SIK**	The machine manufacturer can enable all software options once for a period of 90 days (e.g., for testing). The control indicates the status of the general key: ■ **NONE** The general key has not yet been used for this software version. ■ **dd.mm.yyyy** Date up to which all software options will be available. Once the general key has expired, it cannot be used again. ■ **EXPIRED** The general key has expired for this software version.
Enabling of options Only with **SIK2**	You can select a key file in order to enable software options. You can obtain the key file from your machine manufacturer. **Further information:** "Enabling software options", Page 339
Software Options	The control displays all available options in a table. **Further information:** "The Software Options area", Page 338

The Software Options area

The table with the available software options contains the following columns:

Column	Meaning
#	Number of the software option
Option	Name of the software option On controls with **SIK2**, the part number and the name of the software option are displayed.
	The control indicates the status of the software option by means of the following symbols: ▪ No symbol: The software option is not enabled. ▪ Checkmark: The software option is enabled permanently with all functions. ▪ Clock symbol: The software option has been enabled for a limited period of time or can be ordered again on controls with **SIK2**. ▪ Padlock: The software option has been locked by the machine manufacturer.
Expiration Date or **Status**	The control displays the following information on the status of the software option: ▪ **Enabled** ▪ **YYYY-MM-DD** If a software option has been enabled for a limited period of time, the control shows the date up to which it will be available. ▪ **X of X** On controls with **SIK2**, the control shows how often the software option has been enabled.
Details	Detailed information for the machine manufacturer
Config.	Function that the machine manufacturer can use to lock software options

19.5.1 Viewing of software options

To view enabled software options on the control:

⌂
▶ Select the **Home** operating mode

⚙
▶ Select the **Settings** application
▶ Select **Machine Settings**
▶ Select **SIK**
▶ Navigate to the **Software Options** area
> For enabled software options, the control displays the text **Enabled**.

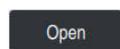## 19.5.2 Enabling software options

To enable a software option for a control with **SIK2**, you need a key file (from the machine manufacturer, for example).

To enable a software option with **SIK2**:

▶ Switch on the power supply of the control and the machine
> The control starts the operating system and displays the **Power interrupted** message.
▶ Navigate to the **SIK** menu item

Select	▶ In the **Enabling of options** area, select the **Select** button > The control opens the **Select key file for options (SIK2_xxx.txt)** window. ▶ Navigate to the folder containing the key file
Open	▶ Select **Open** > The control reads the key file and opens the **Restart** window.
Restart	▶ Select **Restart** > The control restarts and activates the software option.

ⓘ If a key file is present in the **TNC:\SIK2** folder, the control will read that file automatically during the next startup.

Definition

Abbreviation	Definition
SIK (System Identification Key)	**SIK** is the designation of the plug-in board for the control hardware. Each control can clearly be identified by the serial number of the **SIK**. The TNC7 go is equipped with a **SIK2** plug-in board, which means that the numbers of the software options differ from older controls, for example.

19.6 The Machine Times menu item

Application

In the **Machine Times** menu item of the **Settings** application, the control shows the run times since being put into service.

Related topics

- Date and time of the control
  Further information: "The Adjust system time window", Page 345

Description of function

To navigate to this function:

Settings ▶ **Machine Settings** ▶ **Machine Times**

The control displays the following machine times:

Machine time	Meaning
Control On	Run time of the control since being put into service
Machine On	Run time of the machine tool since being put into service
Program Run	Run time of all program runs since being put into service

> ⚙ Refer to your machine manual.
> The machine manufacturer can define up to 20 additional run times.

19.7 Overview of touch probes menu item

Application

The **Overview of touch probes** menu item of the **Settings** application allows you to create and manage all workpiece touch probes and tool touch probes of the control.

Related topics

■ Touch probe table

Further information: "Touch probe table tchprobe.tp (#17 / #1-05-1)", Page 285

■ Creating a tool touch probe with cable or infrared transmission by using the machine parameter **CfgTT** (no. 122700)

Further information: "Machine parameters", Page 391

Description of function

To navigate to this function:

Settings ▶ Machine Settings ▶ Overview of touch probes

The control displays one table for the **SE** transceivers, one table for the **TS** workpiece touch probes, and one table for the **TT** tool touch probes.

The tables contain the following information:

■ Model
■ Number

Only for **TS** and **TT**

■ Serial number
■ **Add** button

Only for **TS** and **TT**

> ⓘ The control shows the table for transceivers only if you use touch probes with radio transmission.

Transceiver unit

If you use touch probes with radio transmission, the control displays the following information for **Transceiver unit**:

Display	Meaning
Status	The transceiver unit is active or inactive
SE	Select the **SE** transceiver
Dimension	Select or change the radio channel
	Select the channel with the best radio transmission and pay attention to overlaps with other machines or handwheels with radio transmission.

Details

If you use touch probes with radio transmission, the control displays the following information for **Details**:

Display	Meaning
Signal strength	The signal strength in the bar chart
	The control shows the currently best-known connection as a complete bar
Deflection	Stylus deflected or not deflected
Collision	Collision or no collision detected
Battery status	If the battery charge falls below the plotted limit, the control displays a warning.

For touch probes with infrared transmission, the control displays the following information for **Details**:

There are no further diagnostic data and functions available for this touch probe.

Buttons

The control displays the following buttons:

Button	Meaning
+	**Add**
	The control adds a new row to the respective table. You define a workpiece touch probe in the Touch probe table and a tool touch probe in the machine parameter **CfgTT** (no. 122700).
Connect	Connect a radio touch probe to a transceiver
Switch-on	Switch touch probe on
Switch off	Switch touch probe off
Change	Change the radio channel used for communication by the touch probe and the transceiver
TNCdiag	The control opens TNCdiag.
	Further information: "TNCdiag", Page 388
Touch probe table	The control opens the Touch probe table.
Delete	The control deletes the highlighted table row.

19.7.1 Connecting a new touch probe with radio transmission

To connect a touch probe with radio transmission:

⚙

> ▶ Select the **Settings** application

> ▶ Select **Machine Settings**
> ▶ Select **Overview of touch probes**
> ▶ Select the desired transceiver unit (e.g., SE 661)
> ▶ Select **Connect new touch probe**
> > The control opens the **Connecting... Please insert the batteries in the touch probe** window.
> ▶ Insert the battery into the touch probe
> > The control closes the window once the touch probe has been connected.
> ▶ Select the newly connected touch probe
> ▶ Select **Switch-on**
> > The status of the touch probe changes to **On**.

19.7.2 Changing the radio channel

Before changing the radio channel, ensure that the touch probe is switched off and that the desired transceiver unit is not connected to any active touch probe.

To change the radio channel:

⚙

> ▶ Select the **Settings** application

> ▶ Select **Machine Settings**
> ▶ Select **Overview of touch probes**
> ▶ Select the desired transceiver unit (e.g., SE 661)
> ▶ Select **Change**
> > The control opens the **Change** window.
> ▶ Select a new channel number
> > The control closes the window and displays the new channel number.

19.8 The Adjustment of analog voltage offset menu item

Application

The **Adjustment of analog voltage offset** menu item of the **Settings** application allows you to correct the voltage offsets of all analog axes that are present.

> ⚙ Only use this function after consultation with your machine manufacturer.

Requirement

- Machine with analog axes
- If user administration is active, the NC.Setter role
  Further information: "List of roles", Page 455

Description of function

To navigate to this function:

Settings ▶ Machine Settings ▶ Adjustment of analog voltage offset

If there are any analog axes, the control displays a table with the following columns:

Column	Meaning
Axis	All analog axes
In closed control loop	Axis is in a closed control loop or is not in a closed control loop. If the axis is in a closed control loop, the control shows a green checkmark.
Current offset	Currently stored voltage offset
New offset	Currently ascertained voltage offset If permitted by the axis configuration, you can adjust the voltage offset here.
Deviation	Deviation of the ascertained voltage offset from the stored offset

> ℹ If you adjust the voltage offsets in the **New offset** column, the control synchronizes the optional machine parameter **analogOffset** (no. 402810) with the current voltage offsets.

19.9 The Adjust system time window

Application

In the **Adjust system time** window, you can set the time zone, date and time manually or by means of NTP server synchronization.

Related topics

- Run times of the machine tool
  Further information: "The Machine Times menu item", Page 340

Description of function

To navigate to this function:

Settings ▶ Operating System ▶ Date/Time

The **Adjust system time** window contains the following areas:

Area	Function
Set the time manually	Activate this check box to define the following data: ■ Year ■ Month ■ Day ■ Time
Synchronize the time over NTP server	If you activate this check box, the control will automatically synchronize the system time with the defined NTP server. You can add a server with a host name or a URL.
Time zone	You can select your time zone from a list.

19.10 Conversational language of the control

Application

You use the **helocale** window to change the conversational language of the HEROS operating system and the machine parameters to change the NC conversational language of the control's user interface.

The HEROS conversational language only changes after a restart of the control.

Related topics

- Machine parameters of the control

  Further information: "Machine parameters", Page 391

Description of function

To navigate to this function:

Settings ▶ Operating System ▶ Language/Keyboards

You can't define two different conversational languages for the operating system and control.

The **helocale** window consists of the following areas:

Area	Function
Language	Choose the HEROS conversational language from a selection menu
	Only if the machine parameter **applyCfgLanguage** (no. 101305) is defined as **FALSE**.
Keyboards	Select the language layout of the keyboard for HEROS functions

19.10.1 Changing the language

By default, the control assumes the NC conversational language for the HEROS conversational language.

To change the NC conversational language:

▶ Select the **Settings** application

▶ Enter the code number 123

▶ Select **OK**

▶ Select **Machine Parameters**

▶ Double-tap or double-click **MPs for setters**

> The control opens the **MPs for setters** application.

▶ Navigate to the machine parameter **ncLanguage** (no. 101301)

▶ Select the desired language

▶ Select **Save**

> The control opens the **Configuration data changed. All changes.** window.

▶ Select **Save**

> The control opens the notification menu and displays a "Question type" error.

▶ Select **CLOSE CONTROL**

> The control restarts.

> Once the control has restarted, the NC conversational language and the HEROS conversational language are changed.

Notes

■ Use the machine parameter **applyCfgLanguage** (no. 101305) to define whether the control assumes the setting for the NC conversational language for the HEROS conversational language.

   ■ **TRUE** (default): The control assumes the NC conversational language. You can change the language only in the machine parameters.

   Further information: "Changing the language", Page 347

   ■ **FALSE**: The control assumes the HEROS conversational language. You can change the language only in the **helocale** window.

■ Use the optional machine parameter **noRebootDialog** (no. 101306) to define whether, after a change of the conversational language, the control displays a restart message.

19.11 SELinux security software

Application

SELinux is an extension for Linux-based operating systems in the sense of Mandatory Access Control (MAC). The security software protects the system against the execution of unauthorized processes or functions (such as viruses and other malicious software).

The machine manufacturer defines the **SELinux** settings in the **Security Policy Configuration** window.

Related topics

- Security settings with firewall
  Further information: "Firewall", Page 379

Description of function

To navigate to this function:

Settings ▶ Operating System ▶ SELinux

By default, **SELinux** access control is implemented as follows:

- The control executes only programs that are installed with the HEIDENHAIN NC software.
- Safety-relevant files, such as **SELinux** system files or HEROS boot files, may only be modified using explicitly selected programs.
- New files created by other programs may not be run.
- USB data carriers can be deselected.
- Only two processes can run new files:
  - Software update: A software update from HEIDENHAIN can replace or modify system files.
  - SELinux configuration: The configuration of **SELinux** in the **Security Policy Configuration** window is usually protected by a password defined by the machine manufacturer. Please refer to the machine manual.

Note

HEIDENHAIN recommends using **SELinux** as additional protection against attacks from outside the network.

Definition

Abbreviation	Definition
MAC (mandatory access control)	MAC means that the control performs only explicitly permitted actions. **SELinux** is intended as protection in addition to the normal access restriction in Linux. Certain processes and actions can be performed only if the standard functions and access control of **SELinux** permit it.

19.12 Network drives on the control

Application

Use the **Mount Setup** window to connect network drives to the control. If a network drive is connected to the control, the control displays additional drives in the navigation column of the file management.

Related topics

- File management

  Further information: Programming and Testing User's Manual
- Network settings

  Further information: "Ethernet interface", Page 353

Requirements

- Existing network connection
- Control and computer in same network
- Path and access data of drive to be connected are known

Description of function

To navigate to this function:

Settings ▶ Network/Remote Access ▶ Shares

You can define any number of network drives, but only seven can be connected at a time.

The Network drive area

In the **Network drive** area, the control shows a list of all defined network drives, as well as the status of each drive.

The control displays the following buttons:

Button	Meaning
Mount	Connect a network drive
	The control selects the check box in the **Mount** column if an active connection exists.
Unmount	Disconnect a network drive
Auto	Automatically connect the network drive when the control is booting.
	The control selects the check box in the **Auto** column if an active automatic connection exists.
Add	Define a new connection
	Further information: "The Mount assistant window", Page 351
Remove	Delete an existing connection
Copy	Copy connection
	Further information: "The Mount assistant window", Page 351
Edit	Edit the connection settings
	Further information: "The Mount assistant window", Page 351
Private network drive	User-specific connection if user administration is active
	The control selects the check box in the **Privat** column if a user-specific connection exists.

The Status Log area

In the **Status Log** area, the control shows status information and error messages about connections.

Use the **Clear** button to delete the contents of the **Status Log** area.

The Mount assistant window

In the **Mount assistant** window you define the settings for a connection with a network drive.

The **Add**, **Copy** and **Edit** buttons open the **Mount assistant** window.

The **Mount assistant** window contains tabs with the following settings:

Tab	Setting
Drive name	■ **Drive name:** Network drive name in the file management of the control The names must be all uppercase letters, terminated by a colon (**:**). ■ **Volume ID:** Currently no function ■ **Private network drive** With user administration active, the connection is only visible to the user who created it.
Share type	Transfer protocol ■ **Windows share (CIFS/SMB) or Samba server** ■ **UNIX share (NFS)**
Server and Share	■ **Server name:** Server name or IP address ■ **Share name:** Directory accessed by the control
Automount	**Connect automatically (not possible with the "Ask for password?" option)** The control connects the network drive automatically during the starting process.
User name and password (only with Windows share)	■ **Single Sign On** With user administration active, the control automatically connects an encrypted network drive when the user logs in. ■ **Windows user name:** ■ **Ask for password? (not possible with the "Connect automatically" option)** Select whether a password is required upon connecting. ■ **Password** ■ **Password verification**
Mounting options	**Parameters for mount option "-o":** Auxiliary parameters for the connection **Further information:** "Examples of Mounting options", Page 352
Check	The control displays a summary of the defined settings. You can check the settings and save them with **Apply**.

Examples of Mounting options

Enter options without a space, only separated by a comma

Options for SMB

Example	Meaning
domain=xxx	Name of the domain HEIDENHAIN recommends not to include the domain in the user name, but rather specify it as an option.
vers=3.1.1	Protocol version
sec=ntlmssp	Authentication method ntlm Use this option if the control displays the **Permission denied** error message upon connecting.

Options for NFS

Example	Meaning
rsize=8192	Packet size in bytes for data reception Input: **512...8192**
wsize=4096	Packet size in bytes for data transmission Input: **512...8192**
soft,timeo=3	Conditional Mount Time in tenths of a second after which the control will try to connect again
nfsvers=2	Protocol version

> **i** If you use the CIMCO NFS software, you must enter the option nfsvers = 2. CIMCO NFS supports NFS only up to version 2.

Notes

- Have a network specialist configure the control.
- To avoid security gaps, prefer the current versions of the **SMB** and **NFS** protocols.

19.13 Ethernet interface

Application

The control is provided with an Ethernet interface as a standard feature so that you can integrate it into a network.

Related topics

■ Firewall settings
  Further information: "Firewall", Page 379
■ Network drives on the control
  Further information: "Network drives on the control", Page 349
■ External access
  Further information: "The DNC menu item", Page 370

Description of function

The control transfers data via the Ethernet interface using the following protocols:

■ **CIFS** (common internet file system) or **SMB** (server message block)
  The control supports versions 2, 2.1 and 3 of these protocols.
■ **NFS** (network file system)
  The control supports versions 2 and 3 of this protocol.

Connection options

10BaseT / 100BaseTx / 1000BaseTx
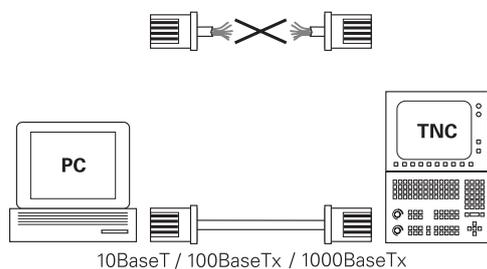
You can integrate the Ethernet interface of the control into the network or connect it directly to a PC through the RJ45 connection X26. The connection is electrically isolated from the control electronics.

> ℹ️ The maximum cable length permissible between the control and a node depends on the quality grade of the cable, the sheathing, and the type of network.

Ethernet connection icon

Icon	Meaning
	Ethernet connection
	The control displays the icon at the bottom right in the taskbar.
	Further information: "Taskbar", Page 434
	When you click the icon, the control opens a pop-up window. The pop-up window contains the following information and functions:
	■ Connected networks
	You can disconnect the network connection. Select the network name to reconnect.
	■ Available networks
	■ VPN connections
	Currently no function

Notes

■ Protect your data and the control by running the machines in a secure network.
■ To avoid security gaps, prefer the current versions of the **SMB** and **NFS** protocols.

19.13.1 The Network settings window

Application

In the **Network settings** window you define the settings for the control's Ethernet interface.

> **i** Have a network specialist configure the control.

Related topics

■ Network configuration
  Further information: "Network configuration with Advanced Network Configuration", Page 448
■ Firewall settings
  Further information: "Firewall", Page 379
■ Network drives on the control
  Further information: "Network drives on the control", Page 349

Description of function

To navigate to this function:

Settings ▶ **Network/Remote Access** ▶ **Network**

Network settings	∧ – □ ×

Status	Interfaces	DHCP server	Ping/Routing	SMB share

Computer name [   ]

Default gateway   10.3.56.254 on eth0   ☐ Use proxy   [Address:Port   ]

Interfaces

Name	Connection	Connection status	Configuration Name	Address
eth0	X26	Activated	DHCP	10.3.56.18
eth1	X116	Activated	DHCP-HostOnly	10.3.56.17

DHCP client

Name	IP address	MAC address	Type	Valid up to

| OK | Apply | OEM authorization | Export configuration | Import the configuration | HEIDENHAIN default | Cancel |

The **Network settings** window

The Status tab

The **Status** tab contains the following information and settings:

Area	Information or Setting
Computer name	The control displays the name under which the control is visible in the company network. You can change the name. **Further information:** "Notes", Page 359
Default gateway	The control shows the default gateway and the Ethernet interface being used.
Use proxy	You can define the **address** and the **port** of a proxy server in the network.
Interfaces	The control shows an overview of available Ethernet interfaces. If there is no network connection, the table is empty. The control displays the following information in the table: ■ **Name** (e.g., **eth0**) ■ **Connection** (e.g., **X26**) ■ **Connection status** (e.g., **CONNECTED**) ■ **Configuration Name** (e.g., **DHCP**) ■ **Address** (e.g., **10.7.113.10**) **Further information:** "The Interfaces tab", Page 357
DHCP client	The control displays an overview of the devices that have received a dynamic IP address in the machine network. If there are no connections to other network components of the machine network, the table is empty. The control displays the following information in the table: ■ **Name** Host name and connection status of the device. The control shows the following connection status: 　■ Green: Connected 　■ Red: No connection ■ **IP address** Dynamically assigned IP address of the device ■ **MAC address** Physical address of the device ■ **Type** Type of connection The control displays the following connection types: 　■ **TFTP** 　■ **DHCP** ■ **Valid up to** Time until which the IP address is valid without being renewed The machine manufacturer can make settings for these devices. Refer to your machine manual.

The Interfaces tab

The control displays the available Ethernet interfaces on the **Interfaces** tab.

The **Interfaces** tab contains the following information and settings:

Column	Information or Setting
Name	The control displays the name of the Ethernet interface. You can activate or deactivate the connection by means of a toggle switch.
Connection	The control displays the number of the network connection.
Connection status	The control displays the connection status of the Ethernet interface. The following connection statuses may be displayed: ■ **CONNECTED** Connected ■ **DISCONNECTED** Connection separated ■ **CONFIGURING** The IP address is being fetched from the server ■ **NOCARRIER** No cable present
Configuration Name	You can execute the following functions: ■ Select a profile for the Ethernet interface In the factory default setting, two profiles are available: ■ **DHCP-LAN**: Settings for the standard interface for a standard company network ■ **MachineNet**: Settings for the second, optional Ethernet interface; for configuration of the machine network **Further information:** "Network configuration with Advanced Network Configuration", Page 448 ■ Reconnect the Ethernet interface with **Reconnect** ■ Edit the selected profile **Further information:** "Network configuration with Advanced Network Configuration", Page 448

> **ⓘ** ■ If you have changed the profile of an active connection, the control will not update the profile being used. Reconnect the corresponding interface with **Reconnect**.
> ■ The control exclusively supports the **Ethernet** connection type.

The DHCP server tab

The machine manufacturer can use the **DHCP server** tab in the control to configure a DHCP server in the machine network. Using this server, the control can establish connections with other network components of the machine network (e.g., with industrial computers).

Refer to your machine manual.

The Ping/Routing tab

You can check the network connection on the **Ping/Routing** tab.

The **Ping/Routing** tab contains the following information and settings:

Area	Information or Setting
Ping	**Address:Port** and **Address:**
	You can enter the IP address of the computer and possibly the port number for checking the network connection.
	Entry: Four numerical values separated by dots and, if necessary, a port number separated by a colon (e.g., **10.7.113.10:22**)
	As an alternative, you can enter the name of the computer whose connection you want to check.
	Starting and stopping the test
	■ **Start** button: starts the test
	The control displays status information in the ping field.
	■ **Stop** button: stops the test
Routing	The control displays status information of the operating system about the current routing for network administrators.

The SMB share tab

The **SMB share** tab is included only in connection with a VBox programming station.

When the check box is active, the control releases areas or partitions protected by a code number for the Explorer of the Windows PC used, e.g. **PLC**. You can activate or deactivate the check box only by using the machine manufacturer code number.

In the **TNC VBox Control Panel**, select a drive letter within the **NC share** tab for displaying the selected partition and then connect the drive with **Connect**. The host displays the partitions of the programming station.

> **ℹ** **Further information:** Programming station for milling controls
>
> You download the documentation together with the programming station software.

Exporting and importing a network profile

To export a network profile:

▶ Open the **Network settings** window
▶ Select **Export configuration**
> The control opens a window.
▶ Select the storage location for the network profile (e.g., **TNC:/etc/sysconfig/ net**)
▶ Select **Open**
▶ Select the desired network profile
▶ Select **Export**
> The control saves the network profile.

ℹ You can't export **DHCP** or **eth1** profiles.

To import an exported network profile:

▶ Open the **Network settings** window
▶ Select **Import the configuration**
> The control opens a window.
▶ Select the storage location of the network profile
▶ Select **Open**
▶ Select the desired network profile
▶ Press **OK**
> The control opens a window with a prompt.
▶ Press **OK**
> The control imports and activates the selected network profile.
▶ You might need to restart the control

ℹ The **HEIDENHAIN default** button allows you to import the default values of the network settings.

Notes

■ Preferably restart the control after making changes in the network settings.
■ If you change the computer name of the control, you must reconfigure the LDAP database of user administration.
  Further information: "Local LDAP database", Page 412
  You must also regenerate the server certificates of **OPC UA NC Server** (#56-61 / #3-02-1*).
  Further information: "Login options", Page 364
■ The HEROS operating system manages the **Network settings** window. You must restart the control in order to change the HEROS conversational language.
  Further information: "Conversational language of the control", Page 346

19.14 PKI Admin

Application

With **PKI Admin**, you can manage the server and client certificates of
OPC UA NC Server (#56-61 / #3-02-1*) on the control. To define access rights to the
control, you can classify the certificates as trusted or not trusted, for example.

PKI Admin has no functionality without the OPC UA NC Server (#56-61 / #3-02-1*)
software option.

Related topics

- Quickly and easily connecting the OPC UA client application to the control
(#56-61 / #3-02-1*)

    Further information: "The OPC UA connection assistant function
(#56-61 / #3-02-1*)", Page 368

Description of function

To navigate to this function:

Settings ▶ Network/Remote Access ▶ PKI Admin

The **Administration of the PKI Infrastructure** window provides the following tabs:

Tab	Function

Own certificates	The control provides the following areas:
	
	■ **Check the configuration**
	
	The control checks the validity of the server certificates.
	
	■ **Use self-created certificate**:
	
	  ■ **Optional certificate settings**
	
	  The control adds static IP addresses to the server certificates. You can select the IP address of the **eth0** or **eth1** interface or specify the required IP addresses.
	
	  ■ **Recreate certificate**
	
	  The control recreates the server's chain of trust. After the next restart of the control, it will use the new certificate.
	
	  ■ **Export certificate chain**
	
	  The control saves the server's chain of trust that you import into the client application.
	
	■ **Use customer-specific certificate**:
	
	  ■ **Load certificate**
	
	  You can import a customized certificate.
	
	  Please note the requirements for self-created certificates for **OPC UA** (#56-61 / #3-02-1*).
	
	  Further information: "Login options", Page 364
	
	■ **Existing certificates of server**
	
	The control displays the available certificates and revocation lists.
	
	You can export the selected certificate or the selected revocation list, show its details, or delete it.

Tab	Function
Trusted	The server knows the certificate and trusts it after successful validation.
	For connection to the server, the client certificate must have been specified on this tab.
	For an **OPC UA** connection (#56-61 / #3-02-1*), you also need to assign an **OPC UA** license to the certificate.
	Further information: "The OPC UA license settings function (#56-61 / #3-02-1*)", Page 369
Issuers	On this tab, you can specify the issuer of the trusted certificates.
	The server uses the issuer's information to validate the certificate.
Rejected	On this tab, the control specifies client certificates whose connection attempt to the **OPC UA NC Server** (#56-61 / #3-02-1*) failed.
	Connection failures can occur in the following situations:
	■ The client certificate is unknown and has not been classified as trusted.
	If you want to connect the client application to the server, you can use the **Move** function to move the certificate to the **Trusted** tab.
	■ A trusted client certificate has expired.
Revocation lists	On this tab, you can specify CRL files that list untrusted certificates.
	The server prohibits connections that use these certificates.
	In the **Settings for revocation lists** area, you can permit connections of applications with certificates in a multi-level certificate chain even if no associated CRL files exist.

Definition

PKI
PKI (public key infrastructure) is the management structure for digital certificates that are required for secure communication. A digital certificate has the same purpose as an identity card or passport. With a digital certificate, its owner can encrypt, sign and authenticate the communication.

19.15 OPC UA NC Server (#56-61 / #3-02-1*)

19.15.1 Fundamentals

Open Platform Communications Unified Architecture (OPC UA) describes 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 secure, reliable, manufacturer- and platform-independent industrial communication over the last years.

In 2016, the German Federal Office for Information Security (BSI) published a security analysis related to **OPC UA**. The security analysis was updated in 2022. The specification analysis performed by the BSI determined that **OPC UA** provides a high level of security as 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 **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.

Application

Both standard and custom software can be used with the **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 **OPC UA NC Server** allows you to access the data and functions of the HEIDENHAIN NC information model exposed in the server address space.

> 📖 Pay attention to the interface documentation of the **OPC UA NC Server** as well as the documentation of the client application.

Related topics

■ **Information Model** interface documentation with the specification of the **OPC UA NC Server** in English
ID: 1309365-xx or **OPC UA NC Server Interface Documentation**

■ Quickly and easily connecting the OPC UA client application to the control
Further information: "The OPC UA connection assistant function (#56-61 / #3-02-1*)", Page 368

■ User roles and user rights for OPC UA
Further information: "User administration roles and rights", Page 455

■ Comparison of the transmission duration of different protocols
Further information: "Example: Transmission duration of different transmission types", Page 445

Requirements

- OPC UA NC Server (#56-61 / #3-02-1*) software options

  For OPC UA-based communication, the HEIDENHAIN control provides the **OPC UA NC Server**. For each OPC UA client to be connected, you need one of the six available software options (56 to 61).

  If your control features the **SIK2**, you can order this software option multiple times and enable up to ten connections.

- Firewall configured

  Further information: "Firewall", Page 379

- The OPC UA client supports the **security policy** and authentication method of **OPC UA NC Server**:
  - **Security Mode: SignAndEncrypt**
  - **Algorithm:**
    - **Basic256Sha256**
    - **Aes128Sha256RsaOaep**
    - **Aes256Sha256RsaPss**
  - **User authentication:**
    - **X509 certificates**
    - User name and password

- For logon with the user name and password:
  - Permitted by the machine manufacturer
  - User administration is active
  - NC.OpcUaPwAuth or NC.OpcUaPwAuthOnlyMachineNet right

Description of function

The control supports the following OPC UA functions:

- Write and read variables
- Subscribe to value changes
- Run methods
- Subscribe to events
- Creation of service files
- Read and write tool data (the corresponding right is required)
- Read from and write to the counter (the corresponding right is required)
- File system access to the **TNC:** drive
- File system access to the **PLC:** drive (the corresponding right is required)
- Validation of 3D models for tool carriers

  Further information: "Tool carrier management", Page 165

Machine parameters in conjunction with OPC UA

The **OPC UA NC Server** enables OPC UA client applications to 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)

  Further information: "The Machine information area", Page 336

- For the machine tool manufacturer: **CfgOemInfo** (no. 131600)

  Further information: "The Info about machine manufacturer area", Page 336

Access to directories

The **OPC UA NC Server** enables read and write access to the **TNC:** and **PLC:** drives.

The following actions are permitted:

- Creating and deleting folders
- Reading, editing, copying, moving, creating, and deleting files

While the NC software is running, the files referenced in the following machine parameters are locked against write access:

- Tables referenced by the machine manufacturer in the machine parameter **CfgTablePath** (no. 102500)
- Files referenced by the machine manufacturer in the machine parameter **dataFiles** (no. 106303, branch **CfgConfigData** no. 106300)

The **OPC UA NC Server** enables access to the control even if the NC software is switched off. As long as the operating system is active, you can create and transmit service files, for example.

NOTICE

Caution: potential damage to property!

The control does not automatically back up the files before editing or deletion. Files that are missing cannot be restored. The removal or editing of system-relevant files, such as the tool table, can negatively affect the control functions.

▶ System-relevant files must be edited only by authorized specialists

---

Login options

The **OPC UA NC Server** requires three different types of certificates. The server and the client need two of them (the application instance certificates) in order to establish a secure connection. The third certificate (user certificate) is required for authorization and for starting a session with specific user permissions. As an alternative to the user certificate, the **OPC UA NC Server** also permits login with a user name and password.

The control automatically generates a two-level certificate chain referred to as the **Chain of Trust** for the server. This certificate chain consists of a self-signed root certificate (including a **revocation list**) and a certificate for the server that is created on the basis of the root certificate.

The client certificate must be added on the **Trusted** tab of the **PKI Admin** function.

All other certificates should be added on the **Issuers** tab of the **PKI Admin** function for verification of the entire certificate chain.

Further information: "PKI Admin", Page 360

User certificate

The control uses the HEROS functions **Current User** or **UserAdmin** for administration of the user certificate. When you initiate a session, the rights of the associated internal user are active.

To assign a user certificate to a user:

▶ Open the **Settings** application

▶ Select **Operating System**

▶ Double-tap or double-click **Current User**

> The control opens the **Active user** window.

▶ Select **SSH keys and certificates**

▶ Select **Import certificate**

> The control opens the **Import certificate** window.

▶ Select the certificate

▶ Select **Open**

> The control imports the certificate.

▶ Select **Use for OPC UA**

> The control uses the certificate for **OPC UA**.

Self-generated certificates

You can also create and import all of the required certificates yourself.

Self-generated certificates must fulfill the following requirements:

- General requirements
  - File format: *.der
  - Signature with hash SHA256
  - Validity period of at most 5 years is recommended
- Client certificates
  - Host name of the client
  - Application URI of the client
- Server certificates
  - Host name of the control
  - Application URI of the server according to the following structure: urn:<hostname>/HEIDENHAIN/OpcUa/NC/Server
  - Validity period of 20 years maximum

Login with user name and password

The machine manufacturer can permit login with a user name and password, for example for client applications that do not support login with a user certificate.

For this type of login, a user with NC.OpcUaPwAuth or NC.OpcUaPwAuthOnlyMachineNet rights must exist while user administration is active.

In the **OPC UA** menu item of the **Settings** application, the control indicates the options available to the current user for logging in.

Further information: "The OPC UA (#56-61 / #3-02-1*) menu item", Page 367

Notes

- OPC UA is a manufacturer/platform-independent, open communication standard. For this reason, an OPC UA client SDK is not included in the **OPC UA NC Server**.
- Refer to your machine manual.

  The machine manufacturer can create additional function users (for example, to enable client applications in order to access specific machine data when user administration is active).

  Further information: "User administration", Page 401

19.15.2 The OPC UA (#56-61 / #3-02-1*) menu item

Application

In the **OPC UA** menu item of the **Settings** application, you can set up the connections to the control and check the status of the **OPC UA NC Server**.

Description of function

To navigate to this function:

Settings ▶ Network/Remote Access ▶ OPC UA

The **OPC UA NC Server** area contains the following functions:

Function	Meaning
Status	Shows with an icon whether the **OPC UA NC Server** is active:
	■ Green icon **OPC UA NC Server** is active
	■ Gray icon: **OPC UA NC Server** is not active or software option not enabled
	You can manually start or restart the **OPC UA NC Server** as required.
	Further information: "Manually starting the OPC UA NC Server", Page 367
OPC UA connection assistant	Open the **OPC UA NC Server connection assistant** window
	Further information: "The OPC UA connection assistant function (#56-61 / #3-02-1*)", Page 368
OPC UA license settings	Open the **OPC UA NC Server - License Settings** window
	Further information: "The OPC UA license settings function (#56-61 / #3-02-1*)", Page 369
PKI Admin	Open the **Administration of the PKI Infrastructure** window
	Further information: "PKI Admin", Page 360
Login options of the current user	The control shows whether the options are available for the current user:
	■ User certificate
	■ User name and password
	Possible only if user administration is active
Host computer operation	Activate or deactivate host computer operation with a toggle switch
	Further information: "The DNC area", Page 370

When a connection is active, the control displays an icon in the information bar.
Further information: "Icons on the control's user interface", Page 73

Manually starting the OPC UA NC Server

You can manually start or restart the **OPC UA NC Server** as required. Thus, you can apply changes made to the machine parameters or the certificates, which are relevant to the server, without having to shut down the control.

While an OPC UA connection is active, the control displays a confirmation prompt before the restart. During the restart, the control will disconnect active connections automatically.

For this function, you need the HEROS.SetNetwork permission.

Further information: "User administration roles and rights", Page 455

19.15.3 The OPC UA connection assistant function (#56-61 / #3-02-1*)

Application

For quick and easy setup of an OPC UA client application, you can use the **OPC UA NC Server connection assistant** window. This assistant guides you through the steps that are required to connect an OPC UA client application to the control.

Related topics

- Assigning the OPC UA client application to a software option 56 to 61 or #3-02-1 to #3-02-10 using the **OPC UA NC Server - License Settings** window

  Further information: "The OPC UA license settings function (#56-61 / #3-02-1*)", Page 369

- Managing certificates with the **PKI Admin** menu

  Further information: "PKI Admin", Page 360

Description of function

Use the **OPC UA** menu item to open the **OPC UA NC Server connection assistant** window.

Further information: "The OPC UA (#56-61 / #3-02-1*) menu item", Page 367

The assistant provides the following steps:

1 Export **OPC UA NC Server** certificates

2 Import the certificates of the OPC UA client application

3 Assign each of the available OPC UA NC Server software options to an OPC UA client application

4 Select the type of user logon: certificate or password

  If you select the user logon with a password, the control will skip the following steps up to the firewall.

5 Import the user certificates

6 Assign the user certificates to users

7 Configure the firewall

8 Connect the client application to the control

If at least one software option is active for the OPC UA NC Server, the control will generate the server certificate as a part of a self-generated certificate chain during the first start-up. The client application or the manufacturer of the application creates the client certificate. The user certificate is linked to the user account. The user name and the password are defined in the user administration. Please contact your IT department.

Note

The **OPC UA NC Server 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.

19.15.4 The OPC UA license settings function (#56-61 / #3-02-1*)

Application

You can use the **OPC UA NC Server - License Settings** window to assign an OPC UA client application to a software option 56 to 61 or #3-02-1 to #3-02-10.

Related topics

- Setting up the OPC UA client application with the **OPC UA connection assistant** function

  Further information: "The OPC UA connection assistant function (#56-61 / #3-02-1*)", Page 368

- Managing certificates with **PKI Admin**

  Further information: "PKI Admin", Page 360

Requirement

- Certificate has been added to the **Trusted** category in **PKI Admin**

Description of function

Use the **OPC UA** menu item to open the **OPC UA license settings** window.

After using the **OPC UA connection assistant** or the **PKI Admin** menu item to import a certificate of an OPC UA client application, you can choose the certificate from a selection window.

If you enable the **Active** check box for a certificate, the control uses a software option for the OPC UA client application.

19.16 The DNC menu item

Application

The **DNC** menu item allows you to permit or block access to the control (e.g., connections via a network or TNCremo).

Related topics

- Connecting network drives
  Further information: "Network drives on the control", Page 349
- Setting up a network
  Further information: "Ethernet interface", Page 353
- TNCremo
  Further information: "PC software for data transfer", Page 438

Description of function

To navigate to this function:

Settings ▶ Network/Remote Access ▶ DNC

The **DNC** area contains the following icons:

Icon	Meaning
✚	**Add** a computer-specific connection
✎	**Edit** a computer-specific connection
🗑	**Delete** a computer-specific connection

When a connection is active, the control displays an icon in the information bar.
Further information: "Icons on the control's user interface", Page 73

The DNC area

In the **DNC** area you use toggle switches to activate the following functions:

Switch	Meaning
DNC access permitted	Permit or block all accesses to the control through a network
TNCopt full access allowed	Permit or block full access via TNCopt Only for testing
Host computer operation	Pass command control to an external host computer, for example to transfer data to the control; or end host computer operation
	If host computer operation is active, the control displays the **Host computer is active** message in the information bar. You cannot use the **Manual** and **Program Run** operating modes.
	You cannot activate host computer operation while running an NC program.

Secure connections

The control displays general and customized settings for **Secure connections**. You can activate the following functions:

Row	Meaning
Fingerprint of the host key	Pressing the **Show** button tells the control to show a unique ASCII image that is equivalent to a fingerprint. When setting up a secure connection, you can compare this ASCII image to an image within the client application. That way you can ensure that you are connecting to the right control.
Setup permitted	If the toggle switch is active, client applications can establish a secure connection for the current user. Activate this toggle switch only while you are setting up a connection.
Key management	In this row, you open the **Certificate and keys** window. **Further information:** "SSH-secured DNC connection", Page 424

Computer-specific connections

If the machine manufacturer has defined the optional machine parameter **CfgAccessControl** (no. 123400), then in the **Connections** area you can permit or block access for up to 32 connections defined by you.

The control shows the defined information in a table:

Column	Meaning
Name	Host name of the external computer
Description	Additional information
IP address	Network address of the external computer
Access	▪ **Permit** The control permits network access without confirmation. ▪ **Inquire** The control asks for confirmation upon a network access attempt. You can choose whether to permit or block access once or always. ▪ **Deny** The control does not permit network access.
Type	▪ **Com1** Serial interface 1 ▪ **Com2** Serial interface 2 ▪ **Ethernet** Network connection
Active	If a connection is active, the control displays a green circle. If a connection is inactive, the control displays a gray circle.

Overview of ports for DNC connections

A DNC connection can use the following protocols:

- SSH (secure connection) with RPC-Secure or LSV2-Secure
- RPC
- LSV2

> ℹ️ When user administration is active, you can set up only secure network connections via SSH or OPC UA (#56-61 / #3-02-1*). If non-secure network connections exist, you must set them up again as secure connections.
>
> If user administration is inactive, the control also automatically blocks non-secure LSV2 or RPC connections. In the optional machine parameters **allowUnsecureLsv2** (no. 135401) and **allowUnsecureRpc** (no. 135402), the machine manufacturer can define whether the control will permit non-secure connections.
>
> **Further information:** "SSH-secured DNC connection", Page 424

Depending on the protocol being used, the connection is established via the following ports:

Port	Control or client PC	Protocol	Firewall
22	Control and client PC with secure connection	RPC-Secure LSV2-Secure	Allow **SSH**
19000	Control	LSV2	Allow **DNC**
19003	Control	RPC	Allow **DNC**
19010-19034	Client PC	RPC	Allow port range

For the client PC, an RPC-Secure connection via an SSH tunnel also uses port 19036. You don't need to allow port 19036, since the secure connection is established via port 22.

Notes

- In the machine parameter **allowDisable** (no. 129202) the machine manufacturer defines whether the **Host computer operation** toggle switch is available.
- In the optional machine parameter **denyAllConnections** (no. 123403) the machine manufacturer defines whether the control permits computer-specific connections.

19.17 Printers

Application

You add and manage printers through the **Printer** menu item in the **Heros Printer Manager** window.

Related topics

- Using the **FN 16: F-PRINT** function for printing

  Further information: Programming and Testing User's Manual

Requirement

- PostScript-capable printer

  The control can communicate only with printers that understand PostScript emulation such as KPDL3. Some printers enable setting PostScript emulation in the printer menu.

  Further information: "Note", Page 375

Description of function

To navigate to this function:

Settings ▶ Network/Remote Access ▶ Printer ▶ Heros Printer Manager

You can print the following files:

- Text files
- Graphic files
- PDF files

Further information: Programming and Testing User's Manual

Once you have added a printer, the control shows the **PRINTER:** drive in the file management. The drive contains one folder for each defined printer.

Further information: "Creating a printer", Page 375

There are various methods to start printing:

- Copying the file to be printed to the **PRINTER:** drive

  The file to be printed is automatically forwarded to the default printer and deleted from the directory after the print job has been executed.

  You may also copy the file into the printer sub-directory if you wish to use a printer other than the default printer.

- Using the **FN 16: F-PRINT** function

Icons and buttons

The **Heros Printer Manager** window provides the following icons and buttons:

Icon or button	Meaning
📄	**Print a test page** Prints a test page on the selected printer
⊗	**Delete** Deletes the selected printer
CREATE	Creates a printer

Icon or button	Meaning
Copy	Creates a copy of the selected printer setting
	At first the copy has the same properties as the copied setting. This can be useful if printing both portrait and landscape formats on the same printer
Status	Displays the status information of the selected printer

The Change the printer window
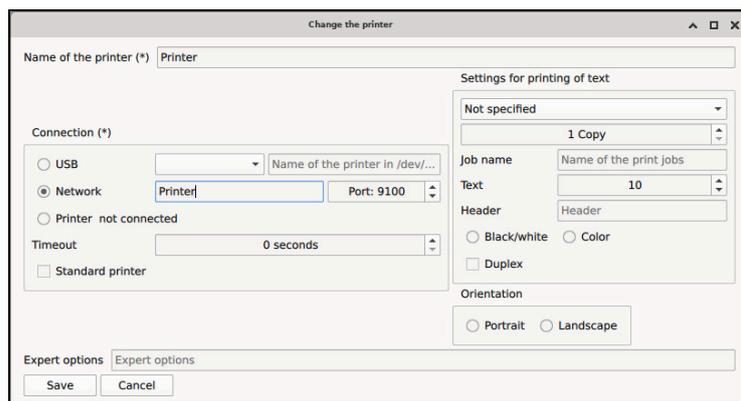
You open the window by double-tapping or double-clicking the desired printer.

For each printer, the following properties can be set:

Area	Meaning
Name of the printer	Customizes the printer name
Connection	■ **USB**: The control automatically displays the name
	■ **Network**: Network name or IP address of the printer
	Port for the network printer (default: 9001)
	■ **Printer %1 not connected**
	■ **Timeout**
	Delays the printing process
	The control delays the printing process by the pre-set number of seconds after the last change has been made to the file to be printed in **PRINTER:**.
	Use this setting if the file to be printed is populated with FN functions (e.g., when probing).
	■ **Standard printer**
	Select the default printer
	The control automatically assigns this setting to the first printer added.

Area	Meaning

Settings for printing of text	■ Paper size
	■ Number of copies
	■ **Job name**
	■ **Font size**
	■ **Header**
	■ Printing options
	  ■ **Black/white**
	  ■ **Color**
	  ■ **Duplex**

Orientation	■ **Portrait** ■ **Landscape**
Expert options	Available only to authorized specialists

19.17.1 Creating a printer

To create a new printer:

▶ Enter the printer name in the input field

▶ Select **CREATE**

> The control creates a new printer.

▶ Double-tap or double-click the printer

> The control opens the **Change the printer** window.

▶ Define the properties

▶ Select **Save**

> The control applies the settings and displays the defined printer in the list.

19.17.2 Copying a printer

To copy the printer properties of an available printer:

▶ Select the desired printer

▶ Enter the name of the new printer in the input field

▶ Select **Copy**

> The control creates a new printer with the settings of the selected printer.

Note

If your printer does not permit PostScript emulation, change the printer settings if possible.

19.18 The VNC menu item

Application

VNC is software that allows you to access the control from remote devices, such as from an additional ITC operating station. **VNC** enables you to transmit screen contents, mouse movements, and keystrokes between devices.

Related topics

*   Firewall settings
    Further information: "Firewall", Page 379

Description of function

To navigate to this function:

Settings ▶ Network/Remote Access ▶ VNC

When an ITC is connected and **Enabling VNC focus** is set, the control displays an icon.

Further information: "The VNC Focus Settings area", Page 378
Further information: "Icons on the control's user interface", Page 73

Icons and buttons

The **VNC settings** window provides the following buttons:

Button	Meaning
Add	Add new VNC viewer or client
Remove	Delete the selected client
	Only possible with manually entered clients.
Edit	Edit the configuration of the selected client
Update	Refresh view
	Required with connection attempts during which the dialog is open.
Set standard values	Reset the settings to their default values
Set preferred owner of the focus	Select the **Preferred owner of the focus** check box

The VNC participant settings area

In the **VNC participant settings** area, the control shows a list of all clients.

The control displays the following contents:

Column	Contents
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

Type	■ Manual
	    Manually entered client
	■ Denied
	    This client is not permitted to connect.
	■ Enable TeleService and IPC
	    Client via a TeleService connection
	■ DHCP
	    Other computer that retrieves an IP address from this computer.

The Firewall warning area

If the firewall blocks **VNC**, the control displays the **Firewall warning** area.

Further information: "Firewall", Page 379

The Global settings area

In the **Global settings** area, you can define the following settings:

Function	Meaning
Enable RemoteAccess and IPC	If the check box is selected, the connection is always permitted.
Password verification	Client must enter a password for verification The control opens a window when you select the check box. In this window you define the password for this client. The client must enter the password when establishing the connection.

The Enabling other VNC area

In the **Enabling other VNC** area, you can define the following settings:

Function	Meaning
Deny	Other VNC clients are not permitted.
Inquire	A dialog opens when another VNC client wants to connect. You must grant permission for this connection.
Permitted	Other VNC clients are permitted.

The VNC Focus Settings area

In the **VNC Focus Settings** area, you can define the following settings:

Function	Meaning
Enabling VNC focus	Enables focus assignment for this system
	When the check box is inactive, the focus owner actively gives away the focus by using the focus symbol. The remaining clients can request the focus only after it was given away.
Reset the CapsLock key when changing the focus	When the check box is active and the focus owner has activated the CapsLock key, the CapsLock key is deactivated if the focus changes.
	Only if the **Enabling VNC focus** check box is enabled
Enable Concurrency VNC Focus	When the check box is active, every client can request the focus at any time. The focus owner does not need to give away the focus before to enable that.
	When a client requests the focus, a pop-up window opens for all clients. If no client objects to the request within the pre-set period of time, the focus changes after the defined time limit.
	Only if the **Enabling VNC focus** check box is enabled
Timeout Concurrency VNC Focus	Period of time after requesting the focus during which the focus owner can object to the focus change (at most 60 seconds).
	This period of time is set by moving a slider. When a client requests the focus, a pop-up window opens for all clients. If no client objects to the request within the pre-set period of time, the focus changes after the defined time limit.
	Only if the **Enabling VNC focus** check box is enabled

> **i** Select the **Enabling VNC focus** check box only in connection with HEIDENHAIN devices provided especially for this purpose.

Notes

- The machine manufacturer defines the procedure for assigning the focus with multiple clients or operating units. Focus assignment depends on the setup and operating situation of the machine tool.
  Refer to your machine manual.
- The control displays a message if the firewall settings of the control do not permit the VNC protocol for all clients.

Definition

Abbreviation	Definition
VNC (virtual network computing)	**VNC** is software that allows you to control other devices over a network connection.

19.19 Firewall

Application

The control provides a firewall to allow or reject incoming network traffic, depending on the sender and service.

Related topics

- Existing network connection

  Further information: "Ethernet interface", Page 353

- SELinux security software

  Further information: "SELinux security software", Page 348

- Ports required for a DNC connection

  Further information: "Overview of ports for DNC connections", Page 372

- Comparison of the transmission duration of different protocols

  Further information: "Example: Transmission duration of different transmission types", Page 445

Description of function

To navigate to this function:

Settings ▶ Network/Remote Access ▶ Firewall
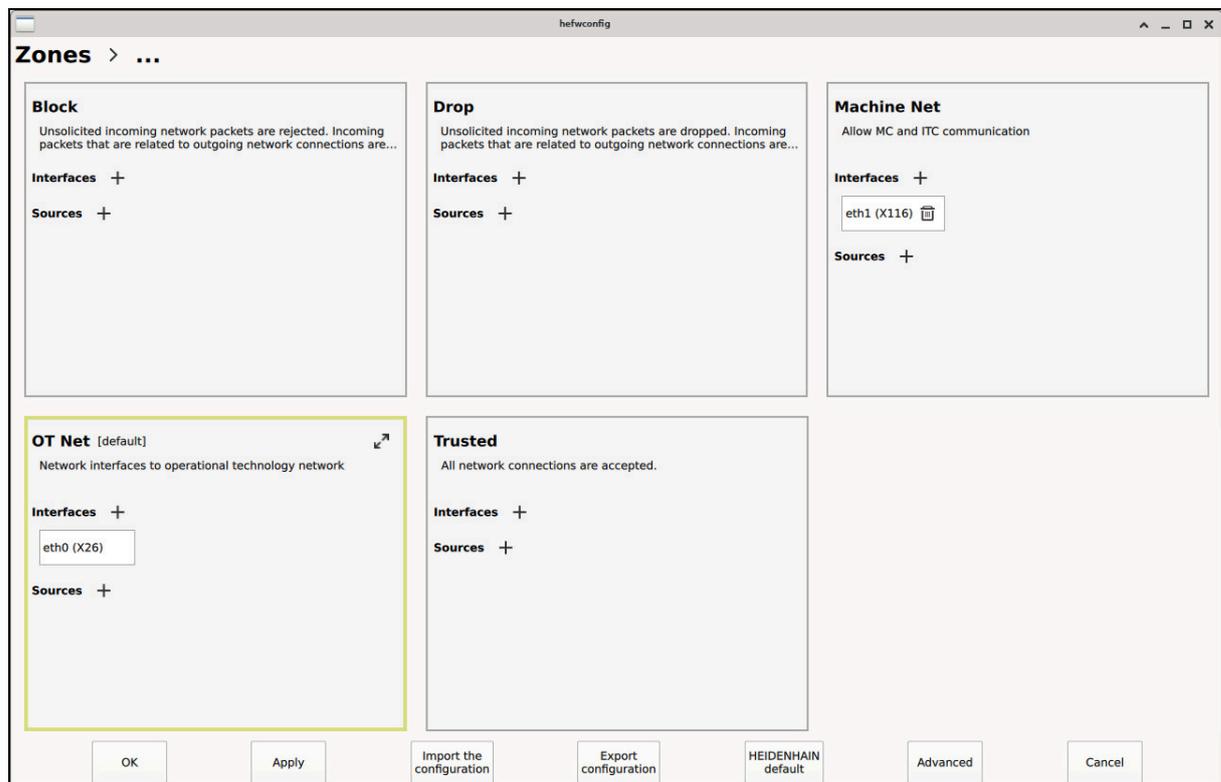
Overview of zones

The **Firewall** window always shows the **OT Net** zone after opening. If you select **Zones** in the breadcrumb navigation, the control opens an overview of the zones.

Five zones are displayed on the default overview screen. The overview shows which interfaces and sources have been added to the respective zones.

Each zone has its own default configuration.

Further information: "Zones", Page 381

You can edit the configuration of the zones.

Further information: "Settings of the zones", Page 382

> ⓘ For example, network specialists can make the following changes:
> - Add and remove zones
> - Rename zones
> - Edit the zone description
> - Edit the default target of the zone
>
> If a network specialist has made changes to the firewall, the firewall of your machine might differ from the default configuration.

Icons and buttons

The **Firewall** window provides the following icons and buttons:

Icon or button	Meaning
⤢	**Maximize** Open the selected zone
⤡	**Reduce** Close the open zone and return to the overview screen
+	**Add**
🗑	**Clear**
✎	**Edit** Edit comprehensive rule
Logging	Activate or deactivate logging of rejected packets in the service file **Further information:** "Service file", Page 203 This information might be needed by HEIDENHAIN Customer Service for diagnostic purposes. Deactivate this function after the fault has been fixed, so as not to impair the performance.
OK	Save the changes and close the window
Apply	Save the changes
Import the configuration	Import the configuration and overwrite the present configuration
Export configuration	Export the configuration of all zones
HEIDENHAIN default	Reset the settings to their default values If the machine manufacturer has stored standard values, the control displays the **OEM default** button
Advanced	Open the **Firewall Configuration** window Available only to network specialists
Cancel	Discard the changes that have not been saved and close the window

Default targets

Each zone has a default target. The default target defines how the firewall will handle incoming network connections. The firewall provides the following default targets:

Default target	Meaning
ACCEPT	Accept all incoming network connections
	This corresponds to the deactivation of the firewall.
DROP	Discard the incoming network connections
	You can add or remove exceptions.
REJECT	Reject the incoming network connections
	You can add or remove exceptions.

Zones

The following table shows the available zones and the default configuration:

Zone	Meaning
Block	Default target: **REJECT**
	This zone rejects all incoming connections.
Drop	Default target: **DROP**
	This zone discards all incoming connections.
Machine Net	Default target: **REJECT** with exceptions
	This zone accepts all the services needed for connections between the control and an additional ITC operating station (e.g., **VNC** or **DNS**).
	The **eth1** interface is assigned to this zone.
OT Net	This zone is the default zone.
	Default target: **REJECT** with exceptions
	This zone accepts the **SSH** service.
	The **eth0** interface is assigned to this zone.
PNC Net	This zone is the default zone on a PNC7.
	Default target: **REJECT** with exceptions
	This zone accepts the services **SSH** and **VNC**.
	On a PNC the **eth0** interface is assigned to this zone.
Trusted	Default target: **ACCEPT**
	This zone accepts all incoming connections.

> ⓘ On programming stations, the **eth1** interface is assigned to the additional zone **Programmingstation Network** by default.

Settings of the zones
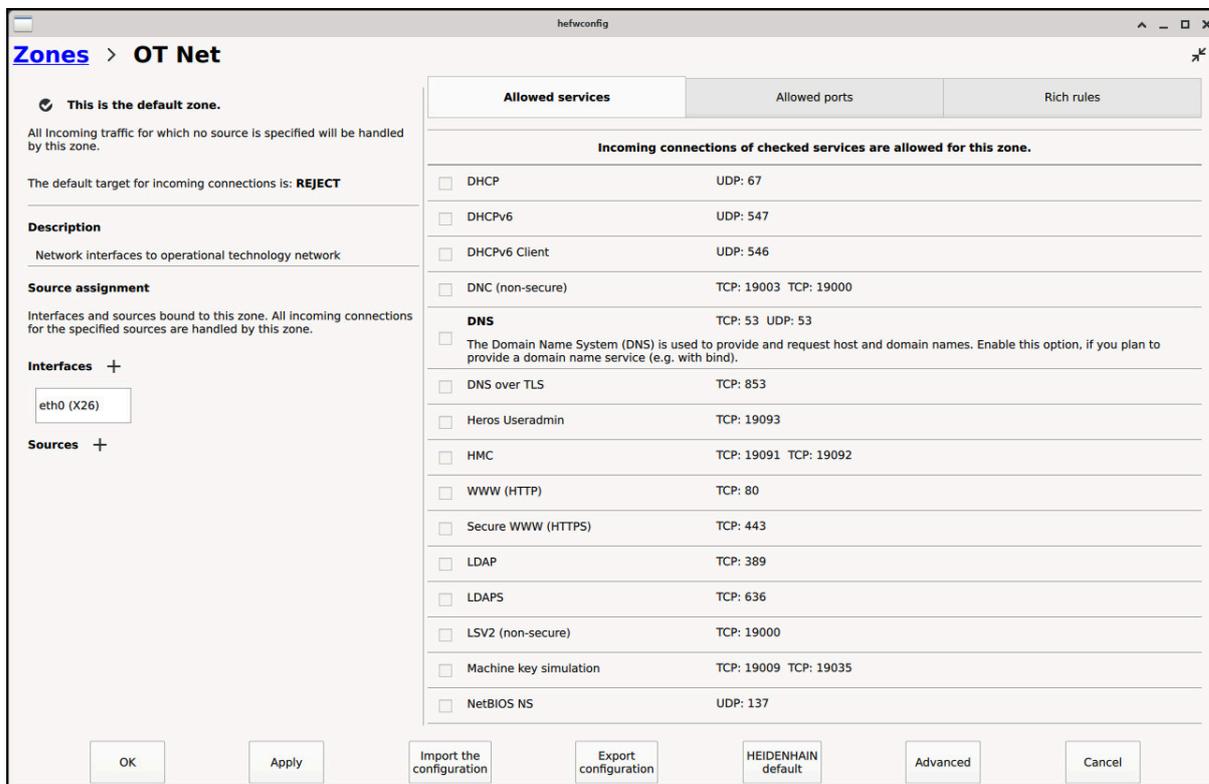
OT Net zone with description of the **DNS** service

When you open a zone, the control displays the following settings:

Setting	Meaning
Default zone	In this area, the control shows whether the zone is the default zone. If the zone is not the default zone, you can define this zone as the default zone by selecting the check box.
	The control automatically assigns all the unassigned interfaces and sources to the default zone.
Source assignment	In this area, the control shows the interfaces and sources assigned to this zone You can add or delete interfaces and sources.
Allowed services	On the **Allowed services** tab, the control displays all available services and the related ports. Use the check boxes to allow or reject services. If the check box is selected, the service is allowed. When you select a service, the control displays the appropriate description.
	ℹ HEIDENHAIN recommends that you add or delete exceptions only in the **OT Net** zone.
Allowed ports	On the **Allowed ports** tab, you can allow the TCP or UDP protocol.
	When you select the **Add** button, the control displays a window. Select TCP or UDP and define the port or the range of ports.

Setting	Meaning
Rich rules	On the **Rich rules** tab, you can define the exceptions for sources, services, and ports in more detail.

When you create a comprehensive rule, the control provides the following selection options:

- **Action**
  - **Accept**

    Accept the selected element
  - **Reject**

    Reject the selected element
  - **Drop**

    Discard the selected element
- **Source**

  IP address or MAC address

  You can also define a rule using the element **Service**, **TCP**, or **UDP** without specifying a source.
- **Element**
  - **All**

    You must specify a source.

    The selected action applies to all services and ports.
  - **Service**

    The control provides a selection menu containing all available services.
  - **TCP**

    The control provides an input field for the port or the range of ports.
  - **UDP**

    The control provides an input field for the port or the range of ports.

Notes

- When user administration is active, you can set up only secure network connections via SSH or OPC UA (#56-61 / #3-02-1*). If non-secure network connections exist, you must set them up again as secure connections.
- You must save all changes by using the **Apply** button; the control will discard the changes that have not been saved.
- You can also open a zone by double-tapping or double-clicking the zone.
- You can assign the interfaces or sources to different zones. A zone will be active once an interface or a source has been assigned to it.
- You can also add or delete interfaces and sources on the overview screen of the zones.
- If you delete an interface or source from a zone, the control will always assign this interface or source to the default zone. You cannot delete any interfaces or sources from the default zone.

19.20 Portscan

Application

With the **Portscan** function, the control checks all open, incoming TCP and UDP listen ports at defined intervals or when commanded. The control shows a message if a port is not listed.

Related topics

- Firewall settings

  Further information: "Firewall", Page 379
- Network settings

  Further information: "Network configuration with Advanced Network Configuration", Page 448

Description of function

To navigate to this function:

Settings ▶ Diagnostics/Maintenance ▶ Portscan

The control searches for all open, incoming TCP and UDP listen ports on the system and compares them to the following whitelists:

- System-internal whitelists **/etc/sysconfig/portscan-whitelist.cfg** and **/mnt/sys/etc/sysconfig/portscan-whitelist.cfg**
- Whitelist for ports with machine-manufacturer-specific functions: **/mnt/plc/etc/sysconfig/portscan-whitelist.cfg**
- Whitelist for ports with customer-specific functions: **/mnt/tnc/etc/sysconfig/portscan-whitelist.cfg**

Each whitelist contains the following information:

- Port type (TCP/UDP)
- Port number
- Offering program
- Comments (optional)

Start the portscan manually by selecting the **Start** button in the **Manual Execution** area. In the **Automatic Execution** area, you can use the **Automatic update on** function to specify that the control will perform the portscan automatically in the selected interval. You define the interval with a slider.

If the control performs the portscan automatically, then only ports listed in the whitelists may be open. The control shows a message window if a port is not listed.

19.21 Backup and restore

Application

The **NC/PLC Backup** and **NC/PLC Restore** functions allow you to back up and restore individual folders or the complete **TNC:** drive. You can save the backup files to various types of memory media.

Related topics

■ File management, **TNC:** drive

Further information: Programming and Testing User's Manual

Description of function

To navigate to this function:

Settings ▶ Diagnostics/Maintenance ▶ NC/PLC Backup

Settings ▶ Diagnostics/Maintenance ▶ NC/PLC Restore

The backup function creates a ***.tncbck** file. The restore function can restore these files as well as files from existing TNCbackup programs. If you double-tap or double-click a ***.tncbck** file in the file manager, the control starts the restore function.

Further information: Programming and Testing User's Manual

Within the backup function you can choose between the following types of backups:

■ **Back up the "TNC:" partition**

Back-up all data on the **TNC:** drive

■ **Back up the directory tree**

Back-up the selected folders and their subfolders on the **TNC:** drive

■ **Back up the machine configuration**

Only for the machine manufacturer

■ **Complete backup (TNC: and machine configuration)**

Only for the machine manufacturer

Backup and restore is subdivided into several steps. Navigate between these steps with the **Continue** and **Back** buttons.

19.21.1 Backing up data

To back-up the data of the **TNC:** drive:

- ▶ Select the **Settings** application

- ▶ Select **Diagnostics/Maintenance**
- ▶ Double-tap or double-click **NC/PLC Backup**
- > The control opens the **Back up the "TNC:" partition** window.
- ▶ Specify the type of backup
- ▶ Select **Continue**
- ▶ If necessary, pause the control with **Stop NC software**
- ▶ Select any predefined exclusion rules or ones you have defined yourself
- ▶ Select **Continue**
- > The control generates a list of files for backing up.
- ▶ Check list
- ▶ Deselect files if necessary

  Further information: "Selecting or deselecting multiple files", Page 387
- ▶ Select **Continue**
- ▶ Enter the name of the backup file
- ▶ Select the storage path
- ▶ Select **Continue**
- > The control generates the backup file.
- ▶ Confirm with **OK**
- > The control concludes the backup process and restarts the NC software.

19.21.2 Restoring data

<table>
<tr><td align="center">NOTICE</td></tr>
<tr><td>Caution: Data may be lost!</td></tr>
<tr><td>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</td></tr>
</table>

To restore data:

⚙️

- ▶ Select the **Settings** application
- ▶ Select **Diagnostics/Maintenance**
- ▶ Double-tap or double-click **NC/PLC Restore**
- > The control opens the **Restore data - %1** window.
- ▶ Select the archive to be restored
- ▶ Select **Continue**
- > The control generates a list of files for restoring.
- ▶ Check list
- ▶ Deselect files if necessary
  Further information: "Selecting or deselecting multiple files", Page 387
- ▶ Select **Continue**
- ▶ If necessary, pause the control with **Stop NC software**
- ▶ Select **Extract archive**
- > The control restores the files.
- ▶ Confirm with **OK**
- > The control restarts the NC software.

19.21.3 Selecting or deselecting multiple files

To select or deselect multiple files at a time:

- ▶ Select the first of the range of desired files.
- ▶ Select **Multiple selection**
- > The control activates multiple selection.
- ▶ Select the last of the range of desired files
- > The control highlights all files starting from the first selected file.
- ▶ Select the check box for selecting or deselecting
- > The control selects or deselects all the highlighted files.
- ▶ Select **Multiple selection**
- > The control deactivates multiple selection.

Note

The TNCbackup PC program can also process ***.tncbck** files. TNCbackup is part of TNCremo.

19.22 TNCdiag

Application

TNCdiag displays status and diagnostic information of HEIDENHAIN components.

Description of function

To navigate to this function:

Settings ▶ **Diagnostics/Maintenance** ▶ **TNCdiag**

> ⚙ Only use **TNCdiag** after consultation with your machine manufacturer, unless you want to use it for setting up wireless handwheels.
> **Further information:** "Setting up a wireless handwheel ", Page 315

> 📖 For general information, please refer to the **TNCdiag** documentation.

19.23 Update the documentation

Application

The **Update the documentation** function can be used, for example, to install or update the integrated **TNCguide** product aid.

Related topics

- Integrated product aid **TNCguide**

  Further information: "User's Manual as integrated product aid: TNCguide", Page 35
- Product aid on the HEIDENHAIN website

  https://content.heidenhain.de/doku/tnc_guide/html/en/

Description of function

To navigate to this function:

Settings ▶ Diagnostics/Maintenance ▶ Update the documentation

The file manager is located in the **Update the documentation** area. You can select and install the desired documentation from the file manager.

Further information: "Transferring TNCguide", Page 390

The control shows all available documents in the **Help** application.

> **i** In the **Update the documentation** area, you can install all HEIDENHAIN-specific documents (e.g., NC error messages).

19.23.1　Transferring TNCguide

To find and transfer the desired **TNCguide** version:

▶ Select the link to the HEIDENHAIN website
　https://content.heidenhain.de/doku/tnc_guide/html/de/index.html

▶ Select **TNC controls**

▶ Select **TNC7 Series**

▶ Select the NC software number

▶ Navigate to the **product aid (HTML)**

▶ Select **TNCguide** in the desired language

▶ Select the path for saving the file

▶ Select **Save**

> The download starts.

▶ Transfer the downloaded file to the control

⌂　　　　　▶ Select the **Home** operating mode

⚙　　　　　▶ Select the **Settings** application

　　　　　　▶ Select **Diagnostics/Maintenance**

　　　　　　▶ Select **Update the documentation**

　　　　　　> The control opens the **Update the documentation** area.

　　　　　　▶ Select the desired file with the extension ***.tncdoc**
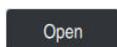
Open　　　　▶ Select **Open**

　　　　　　> The control reports in a window whether installation was
　　　　　　　successful or failed.

⊙　　　　　▶ Select the **Help** application

⌂　　　　　▶ Select **Home**

　　　　　　> The control displays all available documentation.

19.24 Machine parameters

Application

You can configure the behavior of the control with machine parameters. For this purpose, the control provides the **MPs for Users** and **MPs for setters** applications. You can open the **MPs for Users** application at any time without having to enter a code number.

The machine manufacturer defines which machine parameters are in which applications. HEIDENHAIN offers a standard scope of parameters for the **MPs for setters** application. The following contents describe only the standard scope of the **MPs for setters** application.

Related topics

> 📖 **Overview of the Machine Parameters, Error Numbers and System Data**
>
> The additional documentation **Overview of the Machine Parameters, Error Numbers and System Data** provides an overview of the following functions:
>
> - Machine parameters of the **MPs for setters** application
> - Preassigned error numbers of the **FN 14: ERROR** NC function (ISO: **D14**)
> - System data readable with the **FN 18: SYSREAD** (ISO: **D18**) and **SYSSTR** NC functions
>
> ID 1445456-xx
>
> You can download this documentation free of charge from the HEIDENHAIN website.
>
> **TNCguide**

Requirements

- Code number 123

  Further information: "Code numbers", Page 332

- The contents of the **MPs for setters** application have been defined by the machine manufacturer

Description of function

To navigate to this function:

Settings ▶ Machine Parameters ▶ MPs for setters

In the **Machine Parameters** group the control shows only those menu items that you can choose with the current access rights.

If you open an application for machine parameters, the control displays the configuration editor.

The configuration editor offers the following workspaces:

- **Details**
- **Document**
- **List**

You cannot close the **List** workspace.

The configuration editor areas
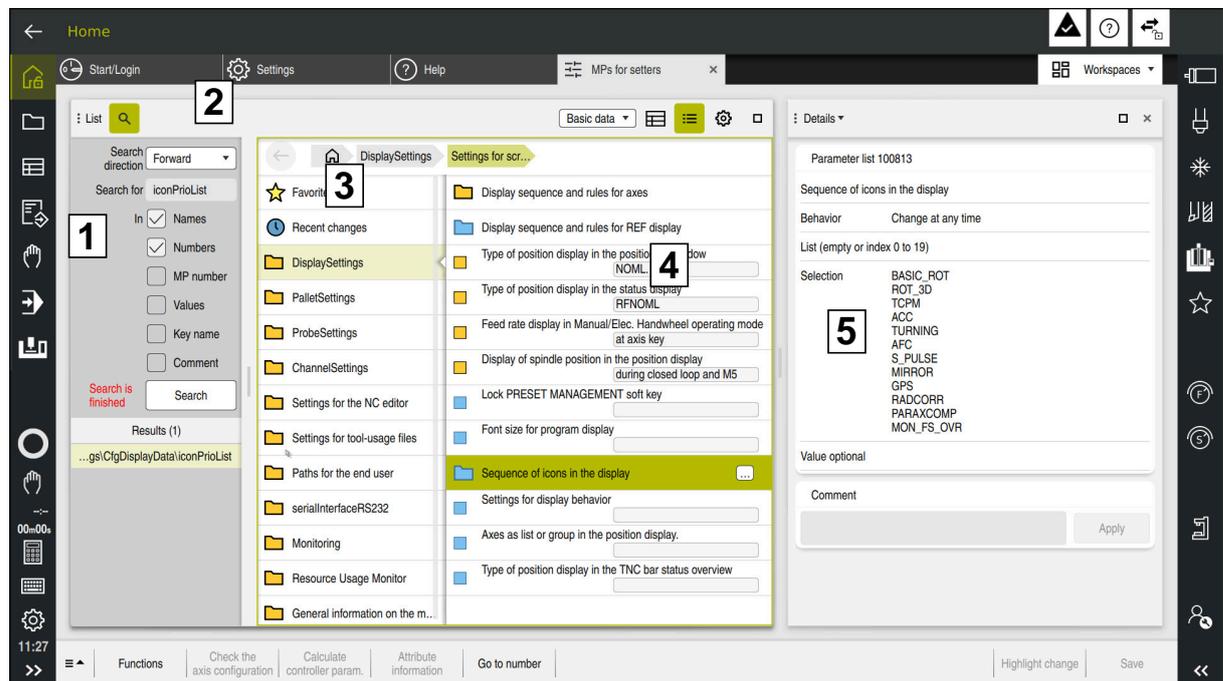
MPs for setters application with selected machine parameter

The configuration editor shows the following areas:

1 The **Search** column

You can search forward or backward with the following characteristics:

- Name

  This is the language-neutral name used for machine parameters in the User's Manual.

- Number

  This is the unique number used for machine parameters in the User's Manual.

- MP number of the iTNC 530

- Value

- Key name

  Machine parameters for axes or channels exist more than once. In order to avoid ambiguity, each axis and each channel is identified with a key name (e.g., **X1**)

- Comment

The control displays the results.

2 Title bar of the **List** workspace

The title bar of the **List** workspace includes the following functions:

- Open or close the **Search** column
- Filter contents using a selection menu
- Toggle between structure and table views

  Further information: "The contents displayed in table view", Page 395

- Open or close the **Details** workspace

  Further information: "The Details workspace", Page 396

- Open or close the **Configuration** window

  Further information: "The Configuration window", Page 396

3　Navigation column

The control provides the following options for navigation:

■ Navigation path

■ Favorites

■ 21 most recent changes

■ Structure of the machine parameters

4　Content column

In the content column the control displays objects, machine parameters, or changes that you select using the search function or navigation column.

5　The **Details** workspace

The control displays information on the selected machine parameter or the most recent change you made.

Further information: "The Details workspace", Page 396

Icons, buttons and shortcuts

The configuration editor provides the following icons, buttons, and shortcuts:

Icon, button, or shortcut	Meaning
⊞	Activate or deactivate the **table view** The control toggles between structure and table views. **Further information:** "The contents displayed in table view", Page 395
‹ǀ›	Active or deactivate the **Change column width** window You can adjust the width of the currently selected column. Only if table view is active
☰	Open or close the **Details** workspace **Further information:** "The Details workspace", Page 396
⚙	Open or close the **Configuration** window **Further information:** "The Configuration window", Page 396
›	**Expand**
⌄	**Collapse** Only if the **Display in tree view** toggle switch is active **Further information:** "The Configuration window", Page 396
🕐	Select **Recent changes**
📁	Object exists ■ Data object ■ Directory ■ Parameter list
📁	Object empty
⌶	**Open context menu** **Further information:** Programming and Testing User's Manual
▮	Machine parameter exists

Icon, button, or shortcut	Meaning
▢	Optional machine parameter does not exist
▢	Machine parameter invalid
🔒✏	Machine parameter readable but not editable
🔒👁	Machine parameter not readable and not editable
✏	Changes to the machine parameter not yet saved
Functions	Open the context menu **Further information:** Programming and Testing User's Manual
[CTRL] + [N]	Create a new object in a list of data objects or parameters
Check the axis configuration	Only for the machine manufacturer
Calculate controller param.	Only for the machine manufacturer
Attribute information	Only for the machine manufacturer
Go to number	The control opens the **Enter number, then select from list** window and suggests the number of the currently selected object. You can enter the number of a machine parameter and navigate directly to this parameter.
Highlight change	The control marks the selected machine parameter as changed, even though the value remains the same. For machine manufacturers only
Save	The control opens a window with all of the changes since the most recent saving. You can save or discard the changes.

The contents displayed in table view

The **List** workspace in table view

The table view of the **List** workspace displays the following information:

1 Group name (key)

The key is displayed in the header of the table. If no key is available, the header is empty.

2 Object name (entity)

The entity is displayed left-aligned in the first column. The name of the entity starts with **Cfg**.

3 Name (attribute) of the machine parameter

The attribute is displayed right-aligned in the first column.

4 Index of the machine parameter

If indexes are available, the index numbers are enclosed within square brackets.

The table view allows you to compare the configurations of different keys, for example.

The Configuration window

The control provides the following toggle switches in the **Configuration** window:

- **Show MP descriptive texts**

  If the toggle switch is active, the control displays a description of the machine parameter in the active conversational language.

  If the toggle switch is not active, the control displays the language-neutral name of the machine parameter.

- **Display in tree view**

  If the toggle switch is active, the control displays the machine parameters in a tree view.

  If the toggle switch is not active, the control displays the machine parameters in the structure view.

- **Do not show attributes without values**

  If the toggle switch is active, the control hides optional machine parameters and parameter lists.

- **Do not show empty folders**

  If the toggle switch is active, the control hides empty data objects and data-object lists.

The Details workspace

If you select contents from the favorites or the structure, the control will display information in the **Details** workspace, such as:

- Type of object, such as data object list or parameter
- Descriptive text of machine parameter
- Permitted or required input
- Prerequisite for the change (e.g., program run blocked)
- Number of the machine parameter on the iTNC 530
- Machine parameter optional

  This information is included if a machine parameter can be enabled optionally.

If you select contents from the most recent changes you made, the control will display the following information in the **Details** workspace:

- Sequential number of the last change
- Previous value
- New value
- Date and time of change
- Descriptive text of machine parameter
- Permitted or required input

Notes

■ The **Home** operating mode icon shown in the TNC bar changes if the code number 123 is enabled. The icon additionally shows an unlocked padlock to illustrate that machine parameters can be accessed.

■ The **CFGREAD** function allows you to read the values from machine parameters. Use the table view to determine the parameters required for **CFGREAD**.

Further information: Programming and Testing User's Manual

■ The machine manufacturer defines which machine parameters are saved user-specifically by the control when user administration is active. These machine parameters can be changed at any time without, for example, having to restart the control.

Further information: "User administration", Page 401

■ The machine manufacturer offers further applications for machine parameters.

■ If later customization of the machine configuration by the machine manufacturer is intended, the machine operator might incur additional costs.

19.25 Configuring the control's user interface

Application

Each user can create and activate configurations in which the control's user interface is customized.

Related topics

■ Workspaces

Further information: "Workspaces", Page 61

■ Control interface

Further information: "Areas of the control's user interface", Page 57

Description of function

To navigate to this function:

Settings ▶ Configurations ▶ Configurations

A configuration contains all adaptations to the control's user interface that do not influence the control's actual functions.

■ Settings for the TNC bar

■ Arrangement of workspaces

■ Font size

■ Favorites

The **Configurations** area contains the following functions:

Function	Meaning
Active Configuration	Activate a configuration from a selection menu **Further information:** "The Desktop menu workspace", Page 76
Default configuration	Use the **Reset** button to apply the settings of the **OEM configuration** to the active configuration.
Save as OEM Configuration	The machine manufacturer can use the **Save** button to overwrite the **OEM configuration**.
Save current settings	With the **Save** button, you can save the current version of the active configuration.
Restore last configuration	With the **Reset** button, you can discard any customizations and revert to the saved version of the active configuration.
Enable autosave	If the toggle switch is active, the control automatically saves changes to the active configuration. If the toggle switch is not active, you can save the active configuration with the **Save current settings** function. If, for example, more than one person is working on the same configuration, the configuration will not be overwritten unintentionally.

The control displays the following information about all available configurations in a table:

Column	Meaning
Configuration Name	Name of the configuration
Selectable	If this toggle switch is active, you can select the configuration in the **Active configuration** selection menu.
Exportable	If this toggle switch is active, you can export the configuration. **Further information:** "Exporting and importing configurations", Page 399
Edit	This column contains two buttons, for renaming and deleting the configuration.

Press the **Add** button to create a new configuration.

19.25.1 Exporting and importing configurations

To export configurations:

⚙️   ▶ Select the **Settings** application

     ▶ Select **Configurations**

     > The control opens the **Configurations** area.

     ▶ Activate the **Exportable** toggle switch for the desired configuration, if necessary
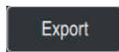
Export   ▶ Select **Export**

     > The control opens the **Save as** window.

     ▶ Select the target file

     ▶ Enter a file name

Create   ▶ Select **Create**

     > The control saves the configuration file.

To import configurations:

Import   ▶ Select **Import**

     > The control opens the **Import configurations** window.

     ▶ Select file

Import   ▶ Select **Import**

     > If importing a configuration would overwrite a file with the same name, the control displays a prompt.

     ▶ Select the procedure:

       ▪ **Overwrite**: The control overwrites the original configuration.

       ▪ **Keep**: The control does not import the configuration.

       ▪ **Cancel**: The control cancels the import process.

Notes

▪ Delete only inactive configurations. If you delete an active configuration, the control first actives a default configuration. This can lead to delays.

▪ The **Overwrite** function permanently replaces existing configurations.

20

User administration

20.1 Fundamentals

Application

User administration enables you to create and administrate different users with different access rights to various functions of the control. You can assign roles to the various users that reflect their respective tasks, such as machine operator or setup technician.

User administration is inactive in the control's factory default setting. This status is called **Legacy-Mode**.

Description of function

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

The user administration differentiates between the following terms:

- User

  Further information: "Users", Page 402
- Roles

  Further information: "Roles", Page 404
- Rights

  Further information: "Rights", Page 404
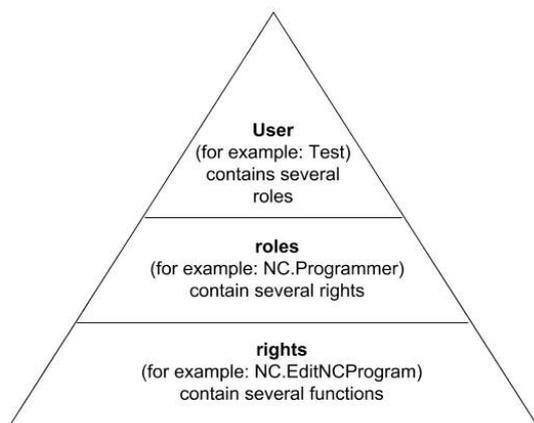
Users

The user administration offers the following types of users:

- Function users pre-defined by HEIDENHAIN
- Function users pre-defined by the machine manufacturer
- Self-defined users

Depending on the task assigned, you can use one of the pre-defined function users or you have to create a new user.

Further information: "Creating a new user", Page 408

If you deactivate user administration, the control saves all configured users. Thus they will be available again when user administration is reactivated.

If you want to delete the configured users upon deactivation, you need to set this explicitly when deactivating user administration.

Further information: "Deactivating user administration", Page 409

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.

- **useradmin**

  The **useradmin** function user is automatically created upon activation of user administration. The **useradmin** function user allows you to configure and edit user administration.

- **sys**

  The **sys** function user allows you to access the **SYS:** drive of the control. This function user is reserved for use by HEIDENHAIN service personnel.

- **user**

  In **legacy mode**, the **user** function user is automatically logged on to the system during control startup. 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**.

- **oem**

  The **oem** function user is intended for the machine manufacturer. The **oem** function user allows you to access the **PLC:** drive of the control.

The useradmin function user

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 user administration

Function users pre-defined by the machine manufacturer

Your machine manufacturer can define up to 32 function users, such as for machine maintenance or for setting up and operating external systems.

Function users defined by the machine manufacturer can also be used as a substitute for code numbers. You can use the function users' passwords to enable their additional rights temporarily.

Further information: "The Active user window", Page 411

The machine manufacturer's function users can already be active in **legacy mode** and replace code numbers.

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.

Further information: "List of roles", Page 455

Advantages of classification in roles:

- Simplified administration
- Different rights are compatible between different software versions of the control and different machine manufacturers.

User administration offers roles for the following tasks:

- **Operating system roles:** access to functions of the operating system and interfaces
- **NC operator roles:** access to functions for programming, setting up and running NC programs
- **Machine tool builder (PLC) roles:** access to functions for configuring and checking the control

Every user should have at least one role from the operating system area and at least one role from the programming area.

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.

Local or remote registration

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.

Further information: "SSH-secured DNC connection", Page 424

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).

You can therefore also make the rights of a user dependent on the access used to operate the control.

Rights

The user administration is based on the Unix rights management. Access to the control is controlled by means of rights.

Rights gather various functions of the control (e.g., editing the tool table).

User administration offers rights for the following tasks:

- HEROS rights
- NC rights
- PLC rights (machine manufacturer)

If more than one role is assigned to a user, he will be granted all rights contained in these roles.

> **i** Ensure that every user is assigned all access rights he needs. The access rights result from the tasks a user performs on the control.

The access rights of HEIDENHAIN function users are already pre-defined in the control's factory default setting.

Further information: "List of rights", Page 458

Password settings

If you use an LDAP database, users with the HEROS.Admin role can define password requirements. For this, the control provides the **Password settings** tab.

Further information: "Saving user data", Page 412

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.

You define the values for each parameter on a scale.

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)

> **i** If you want to use special characters, pay attention to the keyboard layout. HEROS assumes a US keyboard, the NC software assumes a HEIDENHAIN keyboard. External keyboards can be freely configured.

Additional directories

HOME: drive

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 the respectively logged-in users as well as users with the HEROS.Admin role.

public directory

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

The **public** directory can be accessed by any user.

In the **public** directory you can, for example, make files available to other users.

20.1.1   Configuring user administration

User administration needs to be configured before you can use it.

Perform the following steps for configuration:

1   Open the **User administration** window
2   Activating user administration
3   Defining the password for the **useradmin** function user
4   Setting up a database
5   Creating a new user

> **ℹ**   ■   You can exit the **User administration** window after each configuration step.
> ■   If you exit the **User administration** window directly after having activated user administration, the control will prompt you for a restart once.
> ■   When user administration is active, you can set up only secure network connections via SSH or OPC UA (#56-61 / #3-02-1*). If non-secure network connections exist, you must set them up again as secure connections.

Open the User administration window

To open the **User administration** window:

⚙   ▶   Select the **Settings** application

         ▶   Select **Operating System**

         ▶   Double-tap or double-click **UserAdmin**

         >   The control opens the **User administration** window in the **Settings** tab.

            Further information: "The User administration window", Page 410

Activating user administration

To activate user administration:

- ▶ Select **User administration active**
- \> The control shows the message **Enter password for user 'useradmin'**.
- ▶ Retain or reactivate the active status of the **Anonymize users in log data** function

$\mathbf{i}$
- The purpose of the **Anonymize users in log 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.
- If you exit the **User administration** window directly after having activated user administration, the control will prompt you for a restart once.
- When user administration is active, you can set up only secure network connections via SSH or OPC UA (#56-61 / #3-02-1*). If non-secure network connections exist, you must set them up again as secure connections.

Defining the password for the useradmin function user

If you are activating user administration for the first time, you must define a password for the **useradmin** function user.

Further information: "Users", Page 402

To define a password for the **useradmin** function user:

- ▶ Select **Password for useradmin**
- \> The control opens the **Password for user 'useradmin'** pop-up window.
- ▶ Enter the password for the **useradmin** function user

$\mathbf{i}$ Please observe the recommendations for passwords.
Further information: "Password settings", Page 405

- ▶ Repeat the password
- ▶ Select **Set new password**
- \> The control shows the message **Settings were changed**.

Setting up a database

To set up a database:

- ▶ Select the database for saving your user data (e.g., **Local LDAP database**)
- ▶ Select **Configuration**
- \> The control opens a window for configuring the corresponding database.
- ▶ Follow the instructions from the control in the window
- ▶ Select **OK**

$\mathbf{i}$ The following options are available for saving your user data:
- **Local LDAP database**
- **LDAP on remote computer**
- **Connection to Windows domain**

Parallel operation of Windows users and users from an LDAP database is possible.
Further information: "Saving user data", Page 412

Creating a new user

To create a new user:

► Select the **User administration** tab
► Select the **Editing on** checkbox
> The control opens the **Enter password of LDAP database** window.
► Enter password
► Select **OK**
► Select **Create new user**
> The control adds a new user to the **User list**.
► Change the name as needed
► Edit a password as needed
► Define a profile image as needed
► Enter a description as needed
► Select **Add role**
> The control opens the **Add role** window.
► Select a role
► Select **Add**

> **ⓘ**   You can also add roles using the **Add external login** and **Add local login** buttons.
> **Further information:** "Roles", Page 404

► Select **Close**
> The control closes the **Add role** window.
► Select **Apply**
> The control adopts the changes.
► Select **Exit**
> The control opens the **System reboot required** window.
► Select **Yes**
> The control restarts.

> **ⓘ**   The user must change the password when logging in for the first time.

20.1.2   Deactivating user administration

User administration can be deactivated only by the following function users:

- **useradmin**
- **OEM**
- **SYS**

Further information: "Users", Page 402

To deactivate user administration:

▶ Log in as a function user
▶ Open the **User administration** window
▶ Select **User administration inactive**
▶ If desired, check **Delete existing user databases** to delete all configured users and user-specific directories
▶ Select **Apply**
▶ Select **Exit**
> The control opens the **System reboot required** window.
▶ Select **Yes**
> The control restarts.

Notes

NOTICE

Caution: Unwanted data transfer is possible!

If you deactivate the **Anonymize users in log 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 all required data protection provisions have been made at your company.

▶ Retain or reactivate the active status of the **Anonymize users in log data** function

---

- Some user administration areas are configured by the machine manufacturer. Refer to your machine manual.
- HEIDENHAIN recommends activating user administration as part of an IT safety concept.
- If both user administration and a screensaver are active, then the current user's password must be entered to unlock the screen.
  Further information: "HEROS menu", Page 431

20.2 The User administration window

Application

In the **User administration** window you can activate and deactivate user administration, as well as define its settings.

Related topics

- The **Active user** window

  Further information: "The Active user window", Page 411

Requirement

- If user administration is active, the HEROS.Admin role

  Further information: "List of roles", Page 455

Description of function

To navigate to this function:

Settings ▶ Operating System ▶ UserAdmin

The **User administration** window contains the following tabs:

Tab	Meaning
Settings	Configure user administration **Further information:** "Configuring user administration", Page 406
User administration	Create or remove users, change rights, add profile images **Further information:** "Creating a new user", Page 408
Password settings	Define password requirements **Further information:** "Password settings", Page 405

20.3 The Active user window

Application

In the **Active user** window, the control displays information about the logged on user, such as assigned rights. You can also manage other user settings, such as keys for SSH-secured DNC connections or smartcards for logon, and change the password.

Related topics

- SSH-secured DNC connections

  Further information: "SSH-secured DNC connection", Page 424

- Logon with smartcards

  Further information: "Logon with smartcards", Page 420

- Available roles and rights

  Further information: "User administration roles and rights", Page 455

Description of function

To navigate to this function:

Settings ▶ Operating System ▶ Current User

When you open the **Active user** window, by default the window shows the **Base rights** tab. On this tab the control displays information about the user and all assigned roles and rights.

The **Base rights** tab contains the following buttons:

Button	Meaning
Add rights	On the **Added rights** tab, enable rights for another user or function user until the next logoff
Open user administration	Open the **User administration** window **Further information:** "The User administration window", Page 410
SSH keys and certificates	Manage keys and certificates for client connections **Further information:** "SSH-secured DNC connection", Page 424 **Further information:** "OPC UA NC Server (#56-61 / #3-02-1*)", Page 362
Create token	Manage smartcards for logon with a card reader **Further information:** "Logon with smartcards", Page 420
Delete token	
Close	Close the **Active user** window

On the **Change password** tab you can check your password against the current requirements or set a new password.

Further information: "Password settings", Page 405

Note

In legacy mode, the **user** function user is automatically logged on to the system during control startup. When user administration is active, the **user** function user has no effect.

Further information: "Users", Page 402

20.4 Saving user data

20.4.1 Overview

The following options are available for saving your user data:

■ **Local LDAP database**

Further information: "Local LDAP database", Page 412

■ **LDAP on remote computer**

Further information: "LDAP database on a remote computer", Page 413

■ **Connection to Windows domain**

Further information: "Connection to Windows domain", Page 414

> **i** Parallel operation of Windows users and users from an LDAP database is possible.

20.4.2 Local LDAP database

Application

With the **Local LDAP database** setting the control saves the user data locally. That way you can activate user administration even on machines without a network connection.

Related topics

■ Using an LDAP database on multiple controls

Further information: "LDAP database on a remote computer", Page 413

■ Connecting a Windows domain with user administration

Further information: "Connection to Windows domain", Page 414

Requirements

■ User administration is active

Further information: "Activating user administration", Page 407

■ **useradmin** user is logged on

Further information: "Users", Page 402

Description of function

A local LDAP database offers the following options:

■ Using user administration 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

Setting up a Local LDAP database

To set up a **Local LDAP database**:

- ▶ Open the **User administration** window
- ▶ Select **LDAP user database**
- > The control enables the dimmed area for editing the LDAP user database.
- ▶ Select **Local LDAP database**
- ▶ Select **Configuration**
- > The control opens the **Configure local LDAP database** window.
- ▶ Enter the **Name of the LDAP domain:**
- ▶ Enter the password
- ▶ Repeat the password
- ▶ Select **OK**
- > The control closes the **Configure local LDAP database** window.

Notes

- 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.
- If the host name or domain name of the control changes, you need to reconfigure the local LDAP databases.

20.4.3 LDAP database on a remote computer

Application

With the **LDAP on remote computer** function you can transmit the configuration of a local LDAP database between controls and computers. That way you can use the same users on multiple controls.

Related topics

- Configuring an LDAP database on a control
  Further information: "Local LDAP database", Page 412
- Connecting a Windows domain with user administration
  Further information: "Connection to Windows domain", Page 414

Requirements

- User administration is active
  Further information: "Activating user administration", Page 407
- **useradmin** user is logged on
  Further information: "Users", Page 402
- LDAP database has been set up in the company network
- Server configuration file of an existing LDAP database is stored on the control or a PC in the network
  If the configuration file is stored on a PC, the PC must be running and accessible through the network.
  Further information: "Providing a server configuration file", Page 414

Description of function

The **useradmin** function user can export the server configuration file of an LDAP database.

Providing a server configuration file

To provide a server configuration file:
► Open the **User administration** window
► Select **LDAP user database**
> The control enables the dimmed area for editing the LDAP user database.
► Select **Local LDAP database**
► Select **Export server configuration**
> 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 control exports the server configuration file.

Setting up LDAP on remote computer

To set up **LDAP on remote computer**:
► Open the **User administration** window
► Select **LDAP user database**
> The control enables the dimmed area for editing the LDAP user database.
► Select **LDAP on remote computer**
► Select **Import server configuration**
> The control opens the **Import LDAP configuration file window.**
► Select the existing configuration file
► Select **Open**
► Select **Apply**
> The control imports the configuration file.

20.4.4 Connection to Windows domain

Application

With the **Connection to Windows domain** function, you can connect the data of a domain controller with the control's user administration.

Ask your IT administrator to configure the connection to the Windows domain.

Related topics
■ Configuring an LDAP database on a control
  Further information: "Local LDAP database", Page 412
■ Using an LDAP database on multiple controls
  Further information: "LDAP database on a remote computer", Page 413

Requirements
■ User administration is active
  Further information: "Activating user administration", Page 407
■ **useradmin** user is logged on
  Further information: "Users", Page 402
■ Windows domain controller present in the network
■ Domain controller accessible in the network
■ Organizational unit for HEROS roles known
■ Function user is defined in the organization
■ User name and password of the function user are known

Description of function

Your IT administrator sets up a function user for connecting to the Windows domain.

Further information: "Joining a Windows domain", Page 418

Buttons

The **Connection to Windows domain** area provides the following buttons:

Button	Meaning
Configuration	The control opens the **Configure Windows domain with function user** window.
	Further information: "The Configure Windows domain with function user window", Page 416
Find domain	The control selects a Windows domain.
Export the Windows config.	Once you have connected the control to the Windows domain, you can export the configurations for other controls.
	Further information: "Exporting and importing a Windows configuration file", Page 418
Import the Windows config.	Using a present configuration, you can connect the control easily and quickly to the Windows domain.
	Further information: "Exporting and importing a Windows configuration file", Page 418
Check missing role definitions	The control checks whether all of the required roles have been created in the Windows domain.
Add role definition	If any roles required in the Windows domain are missing, you can add the missing roles.
	Further information: "Groups of the domain", Page 417

The Configure Windows domain with function user window

After the domain search, you can customize the Windows domain information or specify new information in the **Configure Windows domain with function user** window.

Your IT administrator will provide the required information.

The **Configure Windows domain with function user** window provides the following settings:

Setting	Meaning
Domain name:	Server name of the Windows domain Is populated by domain search
Key Distribution Center (KDC):	KDC address Is populated by domain search
Alternative admin server:	Deviating server name where the passwords are managed
Map SIDs to Unix UIDs	Map the Windows user SIDs (Security IDs) in Active Directory to the matching Unix UIDs on the control
Use LDAPs	Transfer data using secure LDAPs LDAPs encrypt user data and passwords. You can select a certificate or disable certificate validation.
Group for login authorization:	Define a special group of Windows users to whom you want to restrict the connection to this control
Organizational unit for HEROS roles:	Modify the organizational unit in which the HEROS role names are stored Specify the configuration of your domain.
Prefix for HEROS role names:	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) Is populated by domain search
Separator for HEROS role names:	Modify the separator within the HEROS role names
Function user:	User name and password of the Active Directory function user
Organizational unit for function user:	Organizational unit of the function user
Advanced config-uration of domain section	Only for IT administrators

The function user's user name must not contain blanks. The name and organizational unit form the complete path (Distinguished Name, DN) in the Active Directory.

Groups of the domain

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:

- Use the **Add role definition** function to enter a role directly in the domain
- Use the **Export role definition** function to export the roles to an ***.ldif** file

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 below.

Example 1

The user is a direct or indirect member of the respective group:
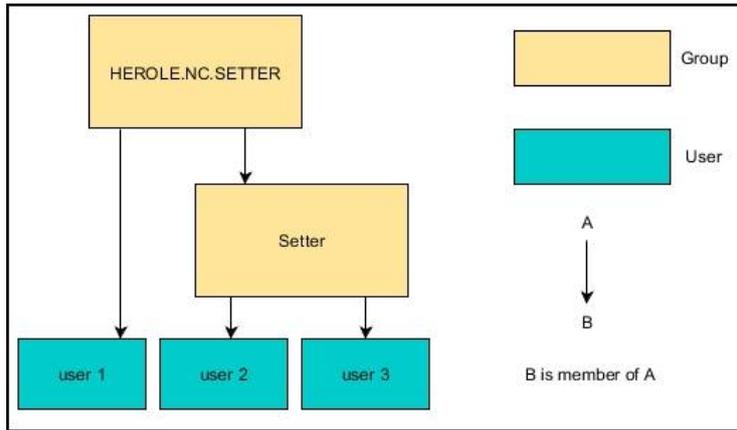
Example 2

Users from various sectors (workshops) are members of groups with different prefixes:
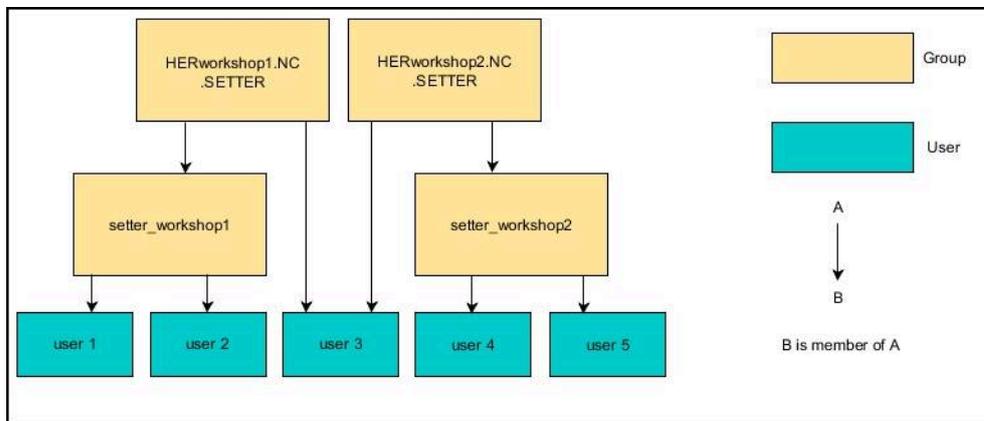
Joining a Windows domain

To join a Windows domain:

► Open the **User administration** window
► Select **Connection to Windows domain**
► Select **Find domain**
> The control selects a domain.
► Select **Configuration**
► Check the data for **Domain name:** and **Key Distribution Center (KDC):**
► Enter **Organizational unit for HEROS roles:**
► Enter the user name and password of the function user
► Press **OK**
► Select **Apply**
> 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.

Exporting and importing a Windows configuration file

If you have connected the control to the Windows domain, you can export the required configurations for other controls.

To export the Windows configuration file:

► Open the **User administration** window
► Select **Connect to Windows domain**
► Select **Export the Windows config.**
> The control opens the **Export the Windows domain configuration** window.
► Select the directory for the file
► Enter the name for the file
► Select the **Export the function user's password?** check box, if required
► Select **Export**
> The control saves the Windows configuration as a BIN file.

To import the Windows configuration file of another control:

► Open the **User administration** window
► Select **Connect to Windows domain**
► Select **Import the Windows config.**
> The control opens the **Import the Windows domain configuration** window.
► Select the existing configuration file
► Select the **Import the function user's password?** check box, if required
► Select **Import**
> The control adopts the configurations for the Windows domain.

20.5 Autologin **with user administration**

Application

If the **Autologin** function is enabled, during startup the control automatically logs on a selected user without the need to enter a password.

As opposed to the **legacy mode**, this enables you to restrict a user's rights without entering a password.

Related topics

■ User login

Further information: "Logging on with user administration", Page 420

■ Configuring user administration

Further information: "Configuring user administration", Page 406

Requirements

■ User administration has been configured

■ The user for **Autologin** has been defined

Description of function

With the **Enable autologin** check box in the **User administration** window, you can define a user for autologin.

Further information: "The User administration window", Page 410

The control then automatically logs this user on and displays the user interface according to the defined rights.

For further authorizations, the control still requires an authentication to be entered.

Further information: "Window for requesting additional rights", Page 423

20.6 Logging on with user administration

Application

The control displays a dialog window for user logon. Within the dialog the user can log on with a password or a smartcard.

Related topics

■ Automatic user logon

Further information: "Autologin with user administration", Page 419

Requirements

■ User administration has been configured
■ For logon with smartcards:
  ■ Euchner EKS card reader
  ■ Smartcard assigned to a user
    Further information: "Assigning a smartcard to a user", Page 421

Description of function

The control displays the Login dialog in the following cases:

■ After the **User logout** function
■ After the **Switch users** function
■ After the **Lock display** function
■ Immediately after control startup if user administration is active and **Autologin** is not enabled

When user administration is active, the control provides these functions in the **Start/Login** application and in the HEROS menu.

Further information: "HEROS menu", Page 431

The logon dialog gives you the following options:

■ Users who logged in at least once
■ **Other** user

Logon with smartcards

You can save a user's logon data on a smartcard and then log the user on with a card reader, without needing to enter a password. You can define whether a PIN is necessary for logon.

The card reader is attached over a USB port. You assign the smartcard to a reader as a token.

Further information: "Assigning a smartcard to a user", Page 421

The smartcard also has additional memory space, where the machine manufacturer can store his own user-specific data.

20.6.1 Logging on a user with password

To logon a user the first time:
- ▶ Select **Other** in the login dialog
- ＞ The control enlarges the user icon you selected.
- ▶ Enter the user name
- ▶ Enter the user's password

> ℹ The control shows in the Login dialog whether CAPS LOCK is active.
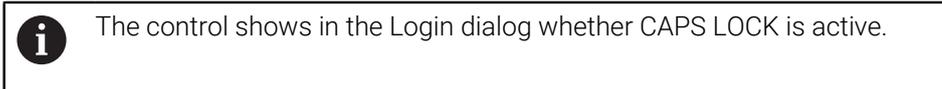
- ＞ The control opens a window with the message **Password expired. Change the password now.**
- ▶ Enter the current password
- ▶ Enter a new password
- ▶ Repeat the new password
- ＞ The control uses the new user to log you in.
- ＞ The control displays this user in the dialog during the next logon procedure.

20.6.2 Assigning a smartcard to a user

To assign a smartcard to a user:
- ▶ Insert a blank smartcard in the card reader
- ▶ Logon the desired smartcard user in user administration

⚙
- ▶ Select the **Settings** application
- ▶ Select **Operating System**
- ▶ Double-tap or double-click **Current User**
- ＞ The control opens the **Active user** window.
- ▶ Select **Create token**
- ＞ The control opens the **Write certificate to token** window.
- ＞ The control displays the smartcard in the **Select token** area.
- ▶ Select the smartcard as the token to be written
- ▶ Activate the **PIN protection?** check box, if desired
- ▶ Enter user password (and PIN, if desired)
- ▶ Select **Start writing**
- ＞ The control saves the user's logon data on the smartcard.
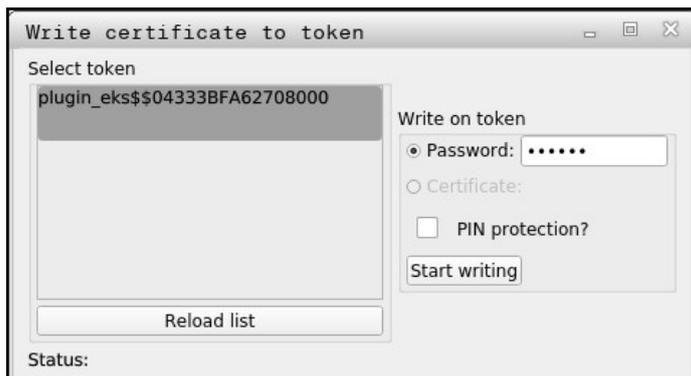
Notes

- You must restart the control in order for it to detect a card reader.
- You can overwrite smartcards that already contain information.
- If you change a user's password, you must reassign the smartcard.

20.7 Window for requesting additional rights

Application

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.

Related topics

- Temporarily granting additional rights in the **Active user** window
  Further information: "The Active user window", Page 411

Description of function

In the **Users that have this right:** field, the control lists all existing users that have the right to use this function.

You must enter the password in order to enable user rights.
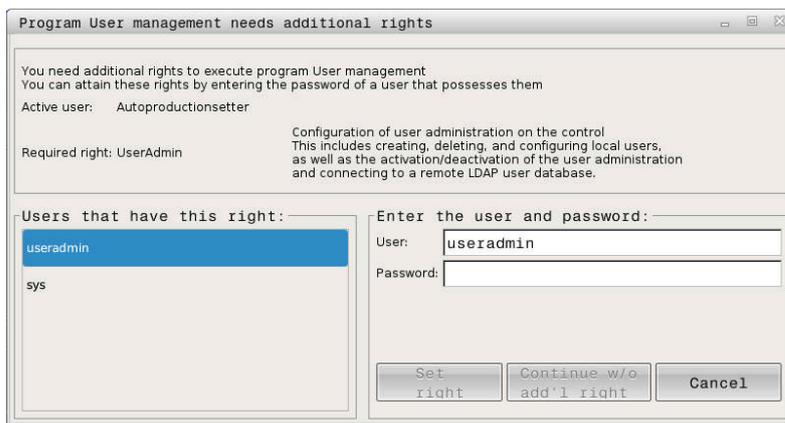
Window for requesting additional rights

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.

Notes

- If **Connection to Windows domain** is used, only users that were recently logged on are shown in the selection menu.
- You can't use this window to change user administration settings. The user with the HEROS.Admin role must be logged on in order to do so.

20.8 SSH-secured DNC connection

Application

If user administration is active, external applications also need to authenticate a user so that the suitable rights can be assigned.

For DNC connections using the RPC or LSV2 protocol, the connection is routed through 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.

Related topics

- Forbidding non-secure connections
  Further information: "Firewall", Page 379
- Roles for remote logon
  Further information: "Roles", Page 404

Requirements

- TCP/IP network
- The remote computer acts as SSH client
- The control acts as SSH server
- Key pair consisting of
  - Private key
  - Public key

Description of function

Concept of transmission through an SSH tunnel

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.
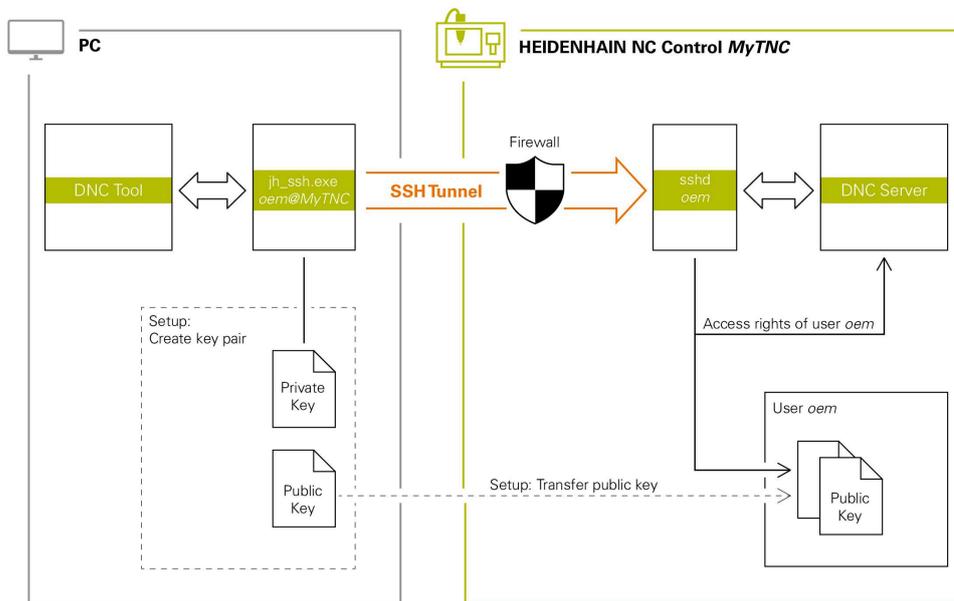
Use in external applications

The PC tools available from HEIDENHAIN, such as TNCremo with version **v3.3** or higher, provide all functions for setting up, establishing, 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.

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.

> **ⓘ** 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.

20.8.1   Setting up SSH-secured DNC connections

To set up an SSH-secured DNC connection for the logged-on user:

{icon}
▶ Select the **Settings** application

▶ Select **Network/Remote Access**
▶ Select **DNC**
▶ Activate the **Setup permitted** toggle switch
▶ Use **TNCremo** to set up the secure connection (TCP secure).

> {icon} For details, refer to the integrated help system of TNCremo.

> TNCremo transmits the public key to the control.

> {icon} In order to ensure maximum security, deactivate the **Allow password authentication** function after the public key has been stored.

▶ Deactivate the **Setup permitted** toggle switch

20.8.2   Removing a secure connection

If you delete a private key from the control, that user no longer has the possibility of a secure connection.

To delete a key:

{icon}
▶ Select the **Settings** application

▶ Select **Operating System**
▶ Double-tap or double-click **Current User**
> The control opens the **Active user** window.
▶ Select **Certificate and keys**
▶ Select the key to be deleted
▶ Select **Delete SSH key**
> The control deletes the selected key.

Notes

- The encryption used with the SSH tunnel protects the communication from attackers.
- For OPC UA connections, a stored user certificate is used for authentication.
  Further information: "OPC UA NC Server (#56-61 / #3-02-1*)", Page 362
- When user administration is active, you can set up only secure network connections via SSH or OPC UA (#56-61 / #3-02-1*). If non-secure network connections exist, you must set them up again as secure connections.

  If user administration is inactive, the control also automatically blocks non-secure LSV2 or RPC connections. In the optional machine parameters **allowUnsecureLsv2** (no. 135401) and **allowUnsecureRpc** (no. 135402), the machine manufacturer can define whether the control will permit non-secure connections.
- Once the connection configurations have been set up, they can be shared among all HEIDENHAIN PC tools for establishing a connection.
- You can also transfer a public key to the control by using a USB device or network drive.
- In the **Certificate and keys** window, you can select a file with additional public SSH keys in the **Externally administered SSH key file** area. This allows you to use SSH keys without having to transfer them to the control.

21

HEROS operating
system

21.1 Fundamentals

HEROS is the fundamental basis for all NC controls from HEIDENHAIN. The HEROS operating system is based on Linux, and was adapted for the purposes of NC controls.

The TNC7 go features the version HEROS 5.

21.2 HEROS menu

Application

In the HEROS menu the control shows information about the operating system. You can change settings or use HEROS functions.

By default you open the HEROS menu through the taskbar at the bottom edge of the screen.

Related topics

■ Opening HEROS functions through the **Settings** application

Further information: "The Settings application", Page 327

Description of function

You open the HEROS menu with the green DIADUR icon in the task bar or with the **DIADUR** key.

Further information: "Taskbar", Page 434
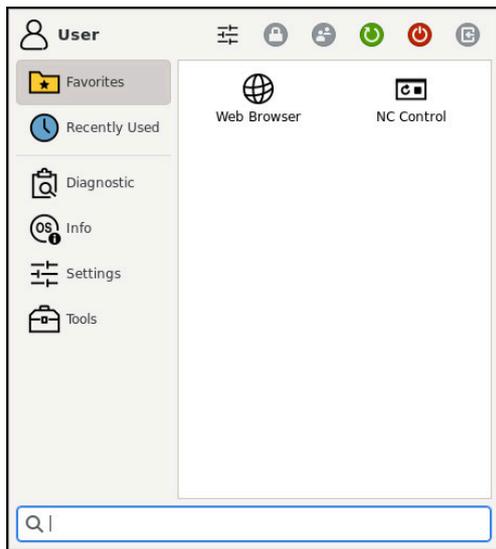
Standard view of the HEROS menu

The HEROS menu contains the following functions:

Area	Function
Header	■ User name
	Further information: "The Active user window", Page 411
	■ User-specific settings
	■ Lock display
	Only if user administration is active
	■ Switch users
	Only if user administration is active
	■ Restart
	■ Shut down
	■ Log out
	Only if user administration is active
	Further information: "User administration", Page 401

Area	Function
Navigation	■ Favorites ■ Recently used
Diagnostic	■ **GSmartControl**: Available only to authorized specialists ■ **HeLogging**: Define settings for internal diagnostic files ■ **ITC VNC**: Display the screen contents of the additional operating station (ITC) ■ **perf2**: Check processor load and process load ■ **Portscan**: Test active connections **Further information:** "Portscan", Page 384 ■ **Portscan OEM**: Available only to authorized specialists ■ **Terminal**: Enter and execute console commands ■ **TNCdiag**: Evaluates status and diagnostic information of HEIDENHAIN components with a focus on the drives and presents it graphically **Further information:** "TNCdiag", Page 388 ■ **TNCscope**: Available only to authorized specialists
Settings	■ **Adjust screen brightness**: Adjust screen brightness ■ **Screensaver**: Screensaver ■ **Current User** **Further information:** "The Active user window", Page 411 ■ **Date/Time** **Further information:** "The Adjust system time window", Page 345 ■ **Firewall** **Further information:** "Firewall", Page 379 ■ **Language/Keyboards** **Further information:** "Conversational language of the control", Page 346 ■ **Network** **Further information:** "Ethernet interface", Page 353 ■ **OEM Function Users** **Further information:** "Users", Page 402 ■ **OPC UA NC Server Connection Assistant** **Further information:** "The OPC UA connection assistant function (#56-61 / #3-02-1*)", Page 368 ■ **OPC UA NC Server License** **Further information:** "The OPC UA license settings function (#56-61 / #3-02-1*)", Page 369 ■ **PKI Admin**: Manage certificates for the control, such as for **OPC UA NC Server** **Further information:** "OPC UA NC Server (#56-61 / #3-02-1*)", Page 362 ■ **Printer** **Further information:** "Printers", Page 373 ■ **Reset Touchscreen Calibration** ■ **Screenshot Config** In the **Screenshot settings** window you can define under which path and file name the control saves screenshots. The file name can contain a placeholder (e.g., %N for sequential numbering).

Area	Function
	■ **SELinux** **Further information:** "SELinux security software", Page 348
	■ **Shares** **Further information:** "Network drives on the control", Page 349
	■ **Touchscreen Calibration**
	■ **Touchscreen Configuration**
	■ **UserAdmin** **Further information:** "The User administration window", Page 410
	■ **VNC** **Further information:** "The VNC menu item", Page 376
	■ **WindowManagerConfig**: Settings for the Window Manager **Further information:** "Window Manager", Page 435
Info	■ **About HeROS**: Open information about the operating system of the control
	■ **About Xfce**: Open information on the Window manager
Tools	■ **Switch-off**: Shut-down or restart
	■ **Screenshot**: Create screenshots
	■ **File Manager**: Available only to authorized specialists
	■ **Document Viewer**: Display and print files (e.g., PDF files)
	■ **Geeqie**: Open, manage, and print graphics
	■ **Gnumeric**: Open, edit, and print tables
	■ **Hostkey**: Show a unique ASCII image to identify the control
	■ **IDS Camera Manager**: Manage cameras connected to the control
	■ **keypad horizontal**: Open virtual keyboard
	■ **keypad vertical**: Open virtual keyboard
	■ **Leafpad**: Open and edit text files
	■ **Meld**: Compare files
	■ **Mozilla Firefox**: Start the browser
	■ **NC Control**: Start or stop the NC software independently of the operating system
	■ **NC/PLC Backup** **Further information:** "Backup and restore", Page 385
	■ **NC/PLC Restore** **Further information:** "Backup and restore", Page 385
	■ **Real VNC Viewer**: Connect to a remote device via a VNC server Available only to network specialists
	■ **Ristretto Image Viewer**: Open graphics
	■ **Secure Remote Access** **Further information:** "Secure Remote Access", Page 442
	■ **Virtual keyboard**
	■ **Touchscreen Cleaning**
	■ **Web Browser**: Start the browser
	■ **Xarchiver**: Extract or compress directories
Search	Full-text search of individual functions

Taskbar

CAD Viewer opened in the third desktop with taskbar shown and active HEROS menu

The taskbar consists of the following areas:

1 Workspaces
2 HEROS menu
Further information: "Description of function", Page 431
3 Opened applications, e.g.:
■ Control interface
■ **CAD Viewer**
■ Window of HEROS functions
You can move the opened applications into any other workspaces.
4 Widgets
■ Calendar
■ Status of the firewall
Further information: "Firewall", Page 379
■ Network status
Further information: "Ethernet interface", Page 353
■ Notifications
■ Shut down or restart the operating system

Window Manager

With the Window Manager, you manage functions of the HEROS operating system as well as windows opened in the third desktop, such as **CAD Viewer**.

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 taskbar 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

If a window is opened in the third desktop, the control displays the **Window Manager** icon in the information bar. You can switch between the open applications by selecting the icon.

You can minimize the control's user interface by pulling down from the information bar. The TNC bar and the OEM bar remain visible.

Further information: "Areas of the control's user interface", Page 57

Notes

- If a window is opened in the third desktop, the control displays an icon in the information bar.

  Further information: "Areas of the control's user interface", Page 57
- The machine manufacturer determines the scope of function and behavior of the window manager.
- 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.

21.3 Serial data transfer

Application

The TNC7 go automatically uses the LSV2 transmission protocol for serial data transfer. All parameters of the LSV2 protocol are invariably fixed except for the baud rate in the machine parameter **baudRateLsv2** (no. 106606).

Description of function

The machine parameter **RS232** (no. 106700) allows you to define another transmission type (interface). The settings described below are effective only for the respective newly defined interface.

Further information: "Machine parameters", Page 391

In the machine parameters that then appear you can define the following settings:

Machine parameters	Setting
baudRate (no. 106701)	Data transfer rate (baud rate)
	Input: **BAUD_110**, **BAUD_150**, **BAUD_300 BAUD_600**, **BAUD_1200**, **BAUD_2400**, **BAUD_4800**, **BAUD_9600**, **BAUD_19200**, **BAUD_38400**, **BAUD_57600**, **BAUD_115200**
protocol (no. 106702)	Data transfer protocol
	▪ **STANDARD**: Standard data transfer, line-by-line
	▪ **BLOCKWISE**: Packet-based data transfer
	▪ **RAW_DATA**: Transfer without protocol (purely character-by-character)
	Input: **STANDARD**, **BLOCKWISE**, **RAW_DATA**
dataBits (no. 106703)	Data bits in each transferred character
	Input: **7 Bit**, **8 Bit**
parity (no. 106704)	Parity bit used to check for transmission errors
	▪ **NONE**: No parity, no error detection
	▪ **EVEN**: Even parity, error if the number of bits set is odd
	▪ **ODD**: Odd parity, error if the number of bits set is even
	Input: **NONE**, **EVEN**, **ODD**
stopBits (no. 106705)	The start bit and one or two stop bits enable the receiver to synchronize to each transmitted character during serial data transmission.
	Input: **1 Stop-Bit**, **2 Stop-Bits**
flowControl (no. 106706)	By handshaking, two devices control data transfer between them. A distinction is made between software handshaking and hardware handshaking.
	▪ **NONE**: No data-flow check
	▪ **RTS_CTS**: Hardware handshaking, transmission stop is active through RTS
	▪ **XON_XOFF**: Software handshaking, transmission stop is active through DC3
	Input: **NONE**, **RTS_CTS**, **XON_XOFF**
fileSystem (no. 106707)	File system for the serial interface
	▪ **EXT**: Minimum file system for printers or non-HEIDENHAIN transmission software
	▪ **FE1**: Communication with TNCserver or an external floppy disk unit
	If you require no special file system, this machine parameter is not needed.
	Input: **EXT**, **FE1**

Machine parameters	Setting
bccAvoidCtrlChar (no. 106708)	The BCC is a block check character. The BCC is optionally added to a transfer block to simplify error detection. ■ **TRUE**: The BCC does not correspond to any control character ■ **FALSE**: Function not active Input: **TRUE**, **FALSE**
rtsLow (no. 106709)	This optional parameter determines the level of the RTS line in the idle state. ■ **TRUE**: Level is **LOW** in idle state ■ **FALSE**: Level is **HIGH** in idle state Input: **TRUE**, **FALSE**
noEotAfterEtx (no. 106710)	This optional parameter sets whether an EOT character (End of Transmission) is to be transmitted after receiving an ETX character (End of Text). ■ **TRUE**: The EOT character is not sent ■ **FALSE**: The EOT character is sent Input: **TRUE**, **FALSE**

Example

In order to use the TNCserver PC software for data transfer, define the following settings in the machine parameter **RS232** (no. 106700):

Parameter	Selection
Data transfer rate in baud	Has to match the setting in TNCserver
Data transfer protocol	BLOCKWISE
Data bits in each transferred character	7 bits
Type of parity checking	EVEN
Number of stop bits	1 stop bit
Type of handshake	RTS_CTS
File system for file operations	FE1

TNCserver is part of the TNCremo software for PCs.

Further information: "PC software for data transfer", Page 438

21.4 PC software for data transfer

Application

HEIDENHAIN offers the TNCremo software for connecting a Windows PC to a HEIDENHAIN control in order to transfer data.

Related topics

- Comparison of the transmission duration of different protocols

    Further information: "Example: Transmission duration of different transmission types", Page 445

Requirements

- PC operating system:
    - Windows 10
    - Windows 11
- PC RAM: 2 GB
- Free PC hard-disk space: 15 MB
- A network connection to the control

Description of function

The TNCremo data transfer software provides the following areas:
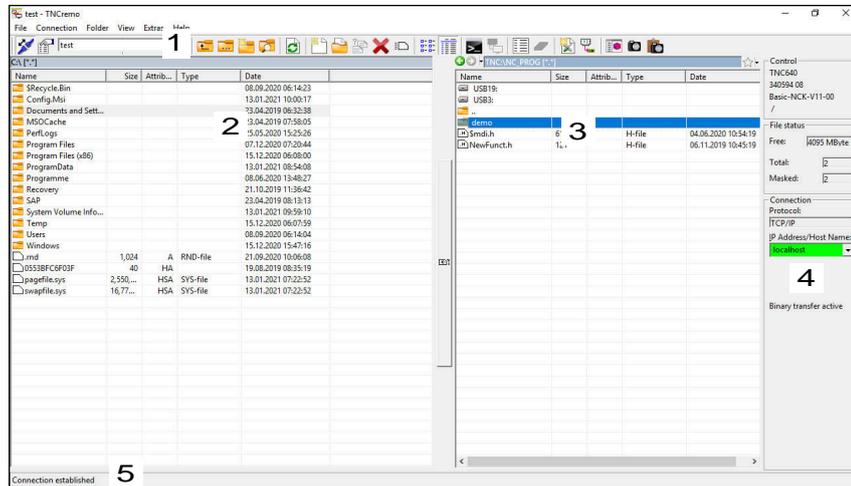
1   Toolbar

This area provides the most important TNCremo functions.

2   File list of the PC

In this area, TNCremo displays all of the folders and files of the connected drive (e.g., hard disk of a Windows PC or a USB flash drive).

3   File list of control

In this area, TNCremo displays all of the folders and files of the connected drive of the control.

4   Status display

In the status display, TNCremo shows information about the current connection.

5   Connection status

The connection status indicates whether a connection is currently active.

> 📖 For more information, refer to the integrated help system of TNCremo.
> You can open the context-sensitive help function of the TNCremo software by pressing the **F1** key.

Notes

- When user administration is active, you can set up only secure network connections via SSH or OPC UA (#56-61 / #3-02-1*). If non-secure network connections exist, you must set them up again as secure connections.

  If user administration is inactive, the control also automatically blocks non-secure LSV2 or RPC connections. In the optional machine parameters **allowUnsecureLsv2** (no. 135401) and **allowUnsecureRpc** (no. 135402), the machine manufacturer can define whether the control will permit non-secure connections.

- You can download the current version of the TNCremo software from the **HEIDENHAIN website** for free.

21.5 File transfer with SFTP (SSH File Transfer Protocol)

Application

SFTP (SSH File Transfer Protocol) provides a secure way to connect client applications to the control and to transfer files at high speed from a PC to the control. The connection is routed via an SSH tunnel.

Related topics

- User administration

  Further information: "User administration", Page 401
- Principle of the SSH connection

  Further information: "Concept of transmission through an SSH tunnel", Page 425
- Firewall settings

  Further information: "Firewall", Page 379
- Comparison of the transmission duration of different protocols

  Further information: "Example: Transmission duration of different transmission types", Page 445

Requirements

- PC software TNCremo with version 3.3 or higher is installed

  Further information: "PC software for data transfer", Page 438
- **SSH** service is permitted in the firewall of the control

  Further information: "Firewall", Page 379

Description of function

SFTP is a secure transmission protocol supported by various operating systems for client applications.

To set up the connection, you need a key pair consisting of a public and a private key. You transfer the public key to the control and assign it to a user through the user administration. The private key is required by the client application to set up a connection to the control.

HEIDENHAIN recommends using the CreateConnections application to generate the key pair. CreateConnections is installed together with the PC software TNCremo with version 3.3 and higher. CreateConnections lets you transfer the public key directly to the control and assign it to a user.

You can also use other software to generate the key pair.

21.5.1   Setting up an SFTP connection with CreateConnections

For an SFTP connection using CreateConnections, the following are required:

- Connection with secure protocol, such as **TCP/IP Secure**
- User name and password of the desired user are known

> ⓘ When you transfer the public key to the control, you must enter the user's password twice.
>
> If user administration is inactive, the user **user** is logged in. The password for the user **user** is **user**.

To set up an SFTP connection:

⚙️

- ▶ Select the **Settings** application

- ▶ Select **Network/Remote Access**
- ▶ Select **DNC**
- ▶ Activate the **Setup permitted** toggle switch
- ▶ Create a key pair with CreateConnections and transfer it to the control

> 📖 For more information, refer to the integrated help system of TNCremo.
>
> You can open the context-sensitive help function of the TNCremo software by pressing the **F1** key.

- ▶ Deactivate the **Setup permitted** toggle switch
- ▶ Transfer the private key to the client application
- ▶ Connect the client application to the control

> 📖 Please refer to the manual of the client application.

Notes

- When user administration is active, you can set up only secure network connections via SSH or OPC UA (#56-61 / #3-02-1*). If non-secure network connections exist, you must set them up again as secure connections. If user administration is inactive, the control also automatically blocks non-secure LSV2 or RPC connections. In the optional machine parameters **allowUnsecureLsv2** (no. 135401) and **allowUnsecureRpc** (no. 135402), the machine manufacturer can define whether the control will permit non-secure connections.

- During the connection, the rights of the user to whom the used key is assigned are active. The directories and files displayed, as well as the access options, vary depending on the permissions.

- You can also transfer a public key to the control by using a USB device or network drive. In this case, you do not need to activate the **Allow password authentication** check box.

- In the **Certificate and keys** window, you can select a file with additional public SSH keys in the **Externally administered SSH key file** area. This allows you to use SSH keys without having to transfer them to the control.

21.6 Secure Remote Access

Application

Secure Remote Access (SRA) allows you to set up an encrypted connection between a PC and your control via the Internet. SRA allows the control to be displayed and operated on a PC, such as for service trainings or remote maintenance.

Related topics

- VNC settings

  Further information: "The VNC menu item", Page 376

Requirements

- Existing Internet connection

  Further information: "Network configuration with Advanced Network Configuration", Page 448

- The following settings in the **VNC settings** window:
  - **Enable RemoteAccess and IPC** check box is active
  - In the **Enabling other VNC** area, the **Inquire** or **Permitted** check box is active

  Further information: "The VNC menu item", Page 376

- PC with paid RemoteAccess software including the extension **Secure Remote Access**

  HEIDENHAIN website

> 📖 For more information, refer to the integrated help system of RemoteAccess.
>
> You can open the context-sensitive help function of the RemoteAccess software by pressing the **F1** key.

Description of function

To navigate to this function:

Tools ▶ Secure Remote Access

The PC provides a ten-digit session ID for you to enter in the **HEIDENHAIN Secure Remote Access** window.

SRA enables connection via a VPN server.
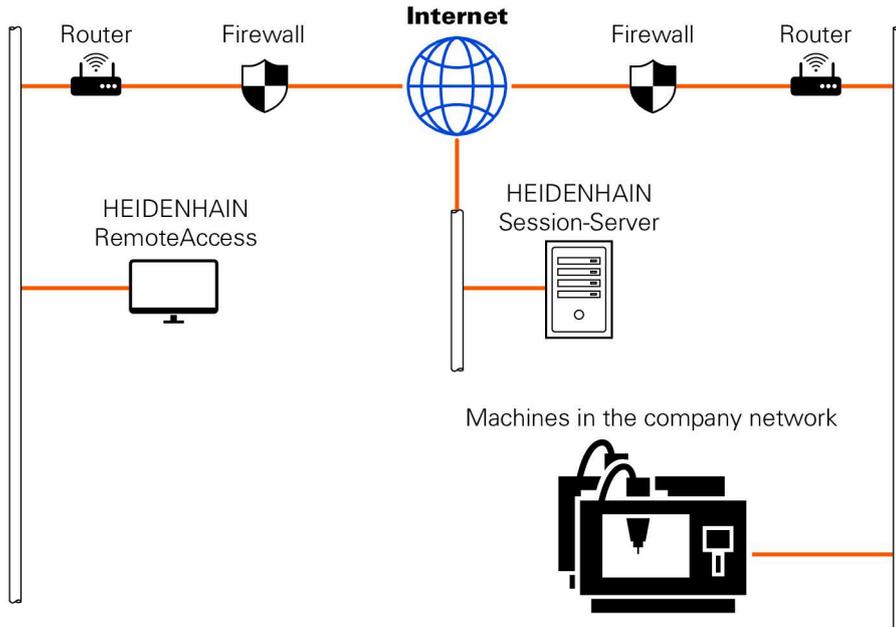
In the **Extended** area, the control shows the progress of the connection setup.

The **HEIDENHAIN Secure Remote Access** window provides the following buttons:

Button	Function
Mount	The control starts the connection with the entered session ID.
Update	The control manually searches for updates for SRA.
	The control automatically searches for available updates when you open the **HEIDENHAIN Secure Remote Access** window.
	If an update is available, you can install it. The control restarts during the update.
Config.	The control opens the **Network settings** window.
	Only for network specialists
Show log	The control opens the log files of the SRA.

Notes

If, in the **VNC settings** window, you set the **Enabling other VNC** setting to **Inquire**, you can permit or deny any connection.

21.7   Data backup

Application

If you create or modify files on the control, then you should back up these files periodically.

Related topics

■ File management

Further information: Programming and Testing User's Manual

Description of function

With the functions **NC/PLC Backup** and **NC/PLC Restore** you can create back-up files for specific directories or even an entire drive, and restore them as needed. You should store these backup files on an external storage medium.

Further information: "Backup and restore", Page 385

You have the following options for transferring files from the control:

■ TNCremo

With TNCremo you can transfer files from the control to a computer.

Further information: "PC software for data transfer", Page 438

■ External drive

You can transfer files from the control directly to an external drive.

Further information: "Network drives on the control", Page 349

■ External data carriers

You can back-up files to external data carriers or use external data carriers to transfer the files.

Further information: Programming and Testing User's Manual

Notes

■ You should back-up all machine-specific data, such as the PLC program or machine parameters. Consult your machine manufacturer about this.

■ You must transmit files with the extensions PDF, XLS, ZIP, BMP, GIF, JPG and PNG in binary format from the PC to the control's hard disk.

■ Backing up all files of the internal memory can take several hours. If required, perform the backup during a time when you don't need the machine.

■ Periodically delete files that are no longer required. This ensures that the control has enough memory available for system files, such as the tool table.

■ HEIDENHAIN recommends having the hard disk inspected after three to five years. After this time, and depending on the operating conditions (e.g., vibration loads), you must expect increased failure rates.

21.7.1 Example: Transmission duration of different transmission types

The following table shows the measurement results regarding how long the different services take to transmit an NC program with a size of 1 GB to and from the control.

A CAD laptop and a TNC7 with an MC 356 main computer were used for the test.

Service	Duration of transmission from the control	Duration of transmission to the control	Further information
TNCremo v3	Approx. 3 min 30 s	Approx. 14 min 30 s	Page 438
TNCremo v4	Approx. 0 min 12 s	Approx. 0 min 12 s	
HEIDENHAIN DNC	Approx. 0 min 12 s	Approx. 0 min 12 s	Page 370
OPC UA NC Server	Approx. 0 min 27 s	Approx. 0 min 50 s	Page 362
SFTP	Approx. 0 min 09 s	Approx. 0 min 10 s	Page 440
SMB	Approx. 0 min 12 s	Approx. 0 min 09 s	Page 353

The duration of transmission can increase due to factors, such as:

- Number of simultaneously transmitted files
- Network utilization
- Age of the hardware being used

21.8 Opening files with additional software

Application

The control provides various tools for opening and editing standard file types.

Related topics

- File types
  Further information: Programming and Testing User's Manual
- Opening image, PDF, and HTML files in the **Document** workspace
  Further information: Programming and Testing User's Manual
- Opening text files in the **Text editor** workspace
  Further information: Programming and Testing User's Manual
- Opening 3D models in **CAD Viewer**
  Further information: "CAD Viewer ", Page 183

Description of function

The control offers tools for the following file types:

File type	Tool
PDF	Document Viewer
XLSX (XLS) CSV	Gnumeric
INI A TXT CFG	Leafpad
CFG	KinematicsDesign
CFT	ToolHolderWizard **Further information:** "Customizing tool carrier templates with ToolHolderWizard", Page 167
HTM/HTML	Web Browser or Mozilla Firefox ℹ️ For networks and the Internet, the machine manufacturer or network administrator must guarantee that the control is protected against viruses and malware (e.g., by a firewall).
ZIP	Xarchiver
BMP GIF JPG/JPEG PNG	Ristretto Image Viewer or Geeqie ℹ️ Ristretto can only open graphics files. Geeqie can also edit and print graphics.
OGG	Parole ℹ️ With Parole you can open the file types OGA, OGG, OGV and OGX. The Fuendo Codec Pack (available for payment) is needed only for other formats, such as MP4 files.

If you double-tap or double-click a file in the file manager, the control automatically starts the file in the correct workspace or tool. If more than one tool is possible for a file, the control displays a selection window.

The control opens the tools in the third desktop.

21.8.1 Opening tools

To open a tool:

▶ Select the HEIDENHAIN icon in the taskbar

> The control opens the HEROS menu.

▶ Select **Tools**

▶ Select the tool (e.g. **Leafpad**)

> The control opens the tool in its own workspace.

Notes

- You can also open several tools from the **Desktop menu** workspace.
- Use the **ALT+TAB** key combination to switch between open workspaces.
- More information on how to use the various tools is provided within the respective tool under Help.
- After starting, the **web browser** checks at regular intervals whether updates are available.

  If you want to update the **web browser**, then you must deactivate the SELinux security software during this time and establish a connection to the Internet. Reactivate SELinux after the update!

  Further information: "SELinux security software", Page 348

21.9 Network configuration with Advanced Network Configuration

Application

Use **Advanced Network Configuration** to edit or remove profiles for the network connection.

Related topics

■ Network settings

Further information: "The Editing network connection window", Page 449

Description of function

When you select the **Advanced Network Configuration** application in the HEROS menu, the control opens the **Network Connections** window.
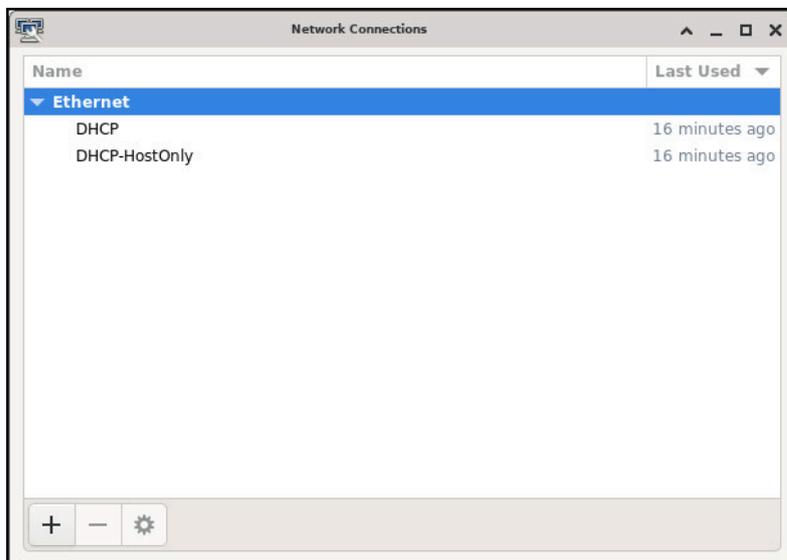
The **Network Connections** window

Icons in the Network Connections window

The following icons are shown in the **Network Connections** window:

Icon	Function
+	Add network connection
–	Remove network connection
⚙	Edit network connection The control opens the **Editing network connection** window. **Further information:** "The Editing network connection window", Page 449

21.9.1 The Editing network connection window

In the **Editing network connection** window, the control shows the connection name of the network connection in the upper area. You can change the name.
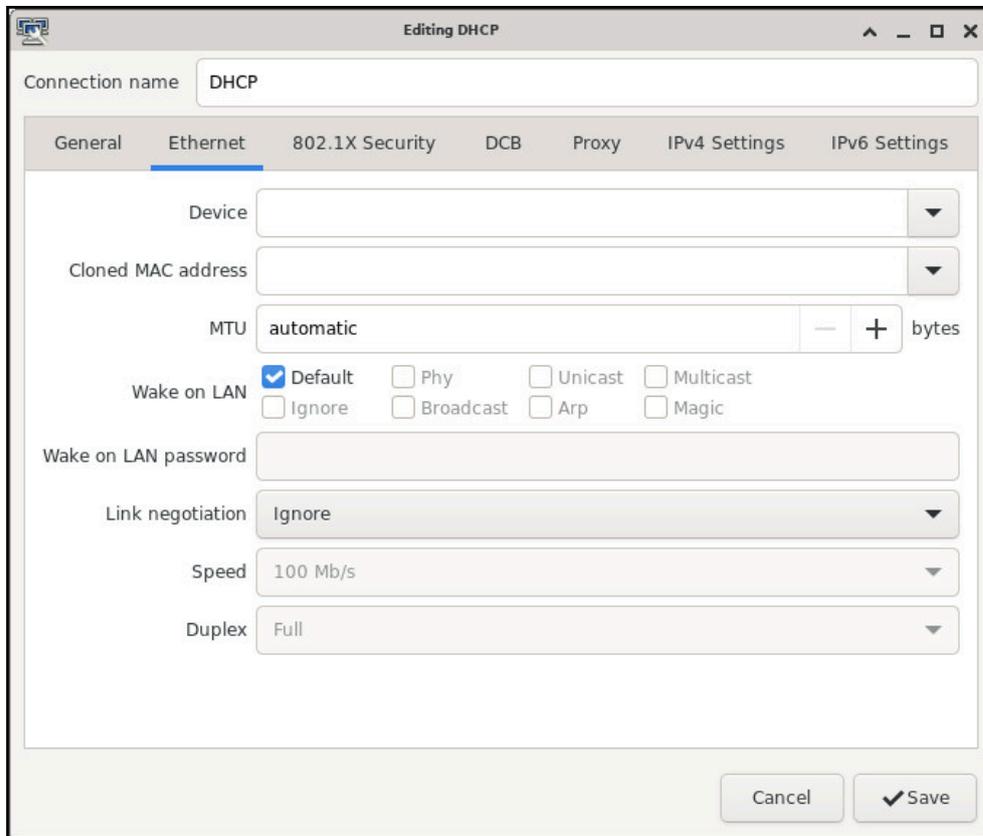
The **Editing network connection** window

The General tab

The **General** tab contains the following settings:

Setting	Meaning
Connect automatically	If you are using several profiles, you can define an order of priority for the connection here.
	The control connects the network with the highest priority first.
	Input: **-999...999**
All users may connect to this network	Here you can enable the selected network for all users.
Automatically connect to VPN when using this connection	Currently no function
Bonded connections:	Currently no function

The Ethernet tab

The **Ethernet** tab contains the following settings:

Setting	Meaning
Service:	Here you can select the Ethernet interface.
	If you do not select an Ethernet interface, this profile can be used for any Ethernet interface.
	Selection by means of a selection window
Cloned MAC address:	Currently no function
MTU:	Here you can define the maximum package size in bytes.
	Input: **Automatic**, **1...10000**
Private key password:	Currently no function
Wake-on-LAN password	Currently no function
Link negotiation	Here you have to configure the settings for the Ethernet connection:
	■ **Ignore**
	Retain the configurations already existing on the device.
	■ **Automatic**
	The speed and duplex settings are configured automatically for the connection.
	■ **Manual**
	Configure the speed and duplex settings for the connection manually.
	Selection by means of a selection window
Speed	Here you have to select the speed settings:
	■ **10 Mb/s**
	■ **100 Mb/s**
	■ **1 Gb/s**
	■ **10 Gb/s**
	Only if **Link negotiation** has been selected **Manual**
	Selection by means of a selection window
Full duplex	Here you have to select the duplex setting:
	■ **Half**
	■ **Full**
	Only if **Link negotiation** has been selected **Manual**
	Selection by means of a selection window

The 802.1X Security tab

Currently no function

The DCB tab

Currently no function

The Proxy tab

Currently no function

The IPv4 Settings tab

The **IPv4 Settings** tab contains the following settings:

Setting	Meaning

Method:	Here you have to select a network connection method:
	■ **Automatic (DHCP)**
	  If the network uses a DHCP server for IP address assignment
	■ **Automatic (DHCP) addresses only**
	  If the network uses a DHCP server for IP address assignment, but you are assigning the DNS server manually
	■ **Manual**
	  Assign the IP address manually
	■ **Link-Local Only**
	  Currently no function
	■ **Shared to other computers**
	  Currently no function
	■ **Disabled**
	  Deactivate IPv4 for this connection

Automatic, addresses only	Here you can add static IP addresses that will be set up in addition to the IP addresses that are assigned automatically. Only with **Method: Manual**
Additional DNS servers:	Here you can add the IP addresses of DNS servers that are used to resolve computer names. Separate multiple IP addresses by commas. Only with **Method: Manual** and **Automatic (DHCP) addresses only**
Additional search domains:	Here you can add domains used by computer names. Separate multiple domains by commas. Only with **Method: Manual**
DHCP client ID:	Currently no function
Require IPv4 addressing for this connection to complete	Currently no function

The IPv6 Settings tab

Currently no function

22

Overviews

22.1 Pin layout and cables for data interfaces

22.1.1 V.24/RS-232-C interface for HEIDENHAIN devices

> **i** The interface complies with the requirements of EN 50178 for Secure separation from the power grid.

Control		25-pin: VB 274545-xx			9-pin: VB 366964-xx		
Male	**Assignment**	**Male**	**Color**	**Female**	**Female**	**Color**	**Female**
1	Do not assign	1	White/Brown	1	1	Red	1
2	RXD	3	Yellow	2	2	Yellow	3
3	TXD	2	Green	3	3	White	2
4	DTR	20	Brown	8	4	Brown	6
5	Signal GND	7	Red	7	5	Black	5
6	DSR	6		6	6	Violet	4
7	RTS	4	Gray	5	7	Gray	8
8	CTR	5	Pink	4	8	White/Green	7
9	Do not assign	8	Violet	20	9	Green	9
Housing	External shield	Housing	External shield	Housing	Housing	External shield	Housing

22.1.2 Ethernet interface RJ45 socket

Maximum cable length:

- 100 m unshielded
- 400 m shielded

Pin	Signal
1	TX+
2	TX−
3	RX+
4	Vacant
5	Vacant
6	RX−
7	Vacant
8	Vacant

22.2 User administration roles and rights

22.2.1 List of roles

> **i** The following contents can change in the following software versions of the control:
> - HEROS role names
> - Unix groups
> - Basic ID number

Further information: "Roles", Page 404

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	■ 335
	■ HEROS.Printer	■ lp	■ 9
	■ HEROS.VMSharedFolders	■ vboxsf	■ 1000
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	■ 334
	■ HEROS.ControlFunctions	■ ctrlfct	■ 340
	■ HEROS.MountUSBDevices	■ mntusb	■ 345
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 NormalUser role, as well as the following rights:		
	■ HEROS.BackupUsers	■ userbck	■ 337
	■ HEROS.PrinterAdmin	■ lpadmin	■ 16
	■ HEROS.ReadLogs	■ logread	■ 342
	■ HEROS.SWUpdate	■ swupdate	■ 341
	■ HEROS.SetNetwork	■ netadmin	■ 336
	■ HEROS.SetTimezone	■ tz	■ 333
	■ HEROS.UserAdmin	■ useradmin	■ 339
	■ HEROS.BackupMachine	■ backup	■ 338

Role	Privileges		
	HEROS role name	**UNIX group**	**Basic ID number**
HEROS.LegacyUser	**Legacy-User** is the default role if user administration is inactive. HEIDEN-HAIN recommends assigning other roles to users when user administration is active.		
	This role grants the rights of the NormalUser role, as well as the following rights:		
	■ HEROS.BackupUsers	■ userbck	■ 337
	■ HEROS.PrinterAdmin	■ lpadmin	■ 16
	■ HEROS.ReadLogs	■ logread	■ 342
	■ HEROS.SWUpdate	■ swupdate	■ 341
	■ HEROS.SetNetwork	■ netadmin	■ 336
	■ HEROS.SetTimezone	■ tz	■ 333
	■ HEROS.UserAdmin	■ useradmin	■ 339
HEROS.LegacyUserNoC-trlfct	This role determines the rights for remote log-in when user administration is disabled (e.g., via SSH). The control assigns this role automatically.		
	This role grants the rights of the LegacyUser role, with the exception of the following right:		
	■ HEROS.ControlFunctions	■ ctrlfct	■ 340

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.OpModeSingleStep	■ NCOpSin-glesStep	■ 303
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	■ NCAp-proveFsAxis	■ 319
	■ NC.EditPocketTable	■ NCEdPocket	■ 307
	■ NC.SetupDrive	■ NCSetupDrv	■ 315
	■ NC.SetupProgramRun	■ NCSe-tupPgRun	■ 303

Role	Privileges		
	HEROS role name	**UNIX group**	**Basic ID number**
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 **LegacyUser** role, the control's behavior regarding NC programming is identical to that of older software versions without user administration. User administration remains active. The **LegacyUser** has the same rights as the AutoProductionSetter.		
	Exception: The **Legacy-User** does not have the NC.ApproveFsAxis right.		
NC.AdvancedEdit	This role allows you to use special functions of the NC and table editors.		
	▪ Special functions for the programming of variables and for editing the table header		
	Replacement for code number **555343**		
	▪ NC.EditNCProgramAdv ▪ NC.EditTableAdv	▪ NCEdit-NCPgmAdv ▪ NCEdit-TableAdv	▪ 327 ▪ 328
NC.RemoteOperator	This role allows you to start NC programs from an external application.		
	▪ NC.RemoteProgramRun	▪ NCRemotePgmRun	▪ 329

Machine manufacturer (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

⊙ Refer to your machine manual.

The machine manufacturer can adapt the PLC roles.

When the **Machine manufacturer (PLC) roles:** are adapted by the machine manufacturer, the following contents may change:
- ▪ The names of the roles
- ▪ The number of roles
- ▪ The functionality of the roles

22.2.2 List of rights

The table below lists all of the individual rights.
Further information: "Rights", Page 404

Rights:

HEROS role name	Description
HEROS.Printer	Data output to network printers
HEROS.PrinterAdmin	Configuration of network printers
HEROS.ReadLogs	Currently no function
NC.OPModeManual	Operation of the machine in the **Manual** operating mode
NC.OPModeMDi	Working in the **MDI** application
NC.OpModeProgramRun	Running NC programs in Full Sequence mode of the **Program Run** operating mode
NC.OpModeSingleStep	Running NC programs in **Single Block** mode of the **Program Run** operating mode
NC.SetupProgramRun	Probing in the **Setup** application
NC.ScheduleProgramRun	Programming a scheduled NC program start
NC.EditNCProgram	Editing NC programs
NC.EditToolTable	Editing the tool table
NC.EditPocketTable	Editing the pocket table
NC.EditPresetTable	Editing the preset table
NC.SetupDrive	Adjustment of drives by the end user
NC.ApproveFsAxis	Confirming test position of safe axes Resetting the test position of the axes
NC.EditNCProgramAdv	Additional NC functions
NC.EditTableAdv	Additional table programming functions (e.g., editing of the table head)
HEROS.SetTimezone	Setting the date and time, time zone and time synchronization via NTP and the **HEROS menu**
HEROS.SetShares	Configuration of public network drives mounted on the control
HEROS.MountShares	Connecting and disconnecting network shares with the control
HEROS.MountUSBDevices	Access to USB devices (e.g., USB flash drive, external hard disk) via the file manager
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

HEROS role name	Description
	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	Defining the NC program start and override values from an external application (e.g., via the DNC interface) Read-access and write-access to the counter by means of **FUNCTION COUNT** via **OPC UA NC Server** (#56-61 / #3-02-1*)
NC.ConfigUserAdv	Configuration access to the contents that have been enabled through code number **123**
NC.DataAccessServiceRead	Read-only access to the **PLC:** drive during servicing
NC.OpcUaOEMConfiguredDataRead	Read-access through **OPC UA NC Server** (#56-61 / #3-02-1*) to data defined by the machine manufacturer
NC.OpcUaOEMConfiguredData	Read-access and write-access through **OPC UA NC Server** (#56-61 / #3-02-1*) to data defined by the machine manufacturer
NC.OpcUaPwAuth	Logon to **OPC UA NC Server** (#56-61 / #3-02-1*) with your user name and password
NC.OpcUaPwAuthOnlyMachineNet	For connection via the **eth1** network interface: Logon to **OPC UA NC Server** (#56-61 / #3-02-1*) with your user name and password

22.3 Overview of signals for Process Tracking Interface (PTI) (#3-04-1*)

> **i** The control always outputs signals in the metric system with Process Tracking Interface (PTI), regardless of the unit of the NC program.

You can output the following signals with PTI:

Signal designator	Description
a act	Actual axis acceleration value [m/s^2] or [°/s^2], calculated via the position encoder. Actual spindle acceleration value [rpm/sec], calculated via the speed encoder.
a nom	Nominal axis acceleration value [m/s^2] or [°/s^2]. Nominal spindle acceleration value [rpm/sec]
v act	Actual axis feed rate [mm/min] or [°/min], calculated via the position encoder. Actual spindle speed value [rpm], calculated via the speed encoder.
v nom	Nominal value of the axis feed rate [mm/min] or [°/min]. Axis feed rate calculated from the difference between the nominal position values. The following error is not included
Feed rate F	Contouring feed rate [mm/min] or [°/min]
Block no.	Block number of the NC program for triggering
s act	Actual position [mm] or [°] with respect to the machine coordinate system M-CS
s nom	Nominal position according to the nominal position value filter [mm] or [°] with respect to the machine coordinate system M-CS
s diff	Following error of the position controller [mm] or [°]
PosDiff	Difference between position and speed encoder [mm] or [°]
j act	Actual jerk value [m/s^3], for rotary axes in [°/s^3]. Calculated via the position encoder.
j nom	Nominal jerk value [m/s^3], for rotary axes in [°/s^3].
GantryDiff	Difference between synchronous axes before and during coupling [mm]
Analog	Analog axis/spindle: Analog voltage = Nominal velocity value [V]
SyncPosDiff	Position difference during spindle synchronism (slave spindle) [°]
AFC Diag.	Diagnosis information for AFC (Adaptive Feed Control, option 45). The following parameters are available: ■ **Spindle utilization** ■ **Override factor** ■ **Cutting condition** ■ **Position** ■ **Status** ■ **Spindle util. accord. to CC** ■ **Load for breakage monitoring**

Signal desig-nator	Description
	■ **Upper overload limit**
	■ **Lower overload limit**
	■ **Idle load of spindle**
	The sampling interval should match the control loop cycle
Thread quality	Thread groove deviates from programmed contour
actTcpWcsX	Current TCP position in X in the workpiece coordinate system
actTcpWcsY	Current TCP position in Y in the workpiece coordinate system
actTcpWcsZ	Current TCP position in Z in the workpiece coordinate system
actTOriwcsZ	Current tool orientation, Euler angle Z
actTOriWcsX'	Current tool orientation, Euler angle X'
actTOriWcsZ''	Current tool orientation, Euler angle Z''
PLCPrePgm	The PLC operands (B, W, D, M, I, O, T, C, IB, IW, ID, OB, OW, OD, s) are recorded **before** the PLC program run. This means that the values of the operands are read at the beginning of the PLC program cycle. For types B, W and D the contents are recorded, and for the other types the logical state of the operands is recorded. For s, a symbolic value can be transferred
PLCPostPgm	The PLC operands (B, W, D, M, I, O, T, C, IB, IW, ID, OB, OW, OD, s) are recorded **after** the PLC program run. This means that the values of the operands are read at the end of the PLC program cycle. For types B, W and D the contents are recorded, and for the other types the logical state of the operands is recorded. For s, a symbolic value can be transferred
Comp	Diagnosis information on non-linear and linear axis-error compensations as well as the effective temperature compensation [mm or °]
kinemComp	Compensation value of KinematicsComp and the kinematic temperature compensation [mm]
tempComp	Pending compensation value of the lag-tracked PLC temperature compensation (PLC Module 9231) [mm or °].
backlashComp	Active compensation of reversal error [mm]
v act vctrl	Actual speed value on the output side, calculated from the motor encoder. Unit of rotary axis: [°/min] Unit of the linear axis: [mm/min]
v nom vctrl	Nominal speed value on the output side at position controller output. Unit of rotary axis: [°/min] Unit of the linear axis: [mm/min]
Iq int	Integral-action component of nominal current value [A]; CC 61xx/CC 3xx: effective value
Iq nom	Torque-defining nominal current value [A]; CC 61xx/CC 3xx: effective value
I2t mot	Current value of the I^2t monitoring of the motor [%]
I2t inv	Current value of the I^2t monitoring of the power module [%]
util-mot	Utilization of a motor with respect to the nominal point [%]

Signal designator	Description
util-DriveTrain	Utilization of the drive train in percent (power module + motor) [%]
J total	Estimated mass moment of inertia [kg * m^2] (for rotary axes) from Load Adaptive Control (LAC—software option 143), if active
m total	Estimated mass [kg] (for linear axes) from Load Adaptive Control (LAC), if active
I act	Momentary actual current of the motor [A]
Id act	Momentary actual magnetizing current of the motor [A]
P mot elec	Electric power consumption of the motor [W]
P UV	Actual power released by the supply module [W]
U nom	Nominal voltage [V]
P mot mech	Mechanical power [W]
M act	Actual torque value [Nm]
P ZK	DC-link power of the power supply module [W]
U DC-link	DC-link voltage [V]
n act	Momentary motor speed [rpm]
n nom	Nominal value of motor speed [rpm]
Brake	Brake is released
s diff Ctrl	Servo lag of the position controller [mm] or [°]
U DC-link (UVR)	UVR DC-link voltage [V]
I DC-link	DC-link current [A]
I L1	Current of phase conductor L1 [A]
I L2	Current of phase conductor L2 [A]
I L3	Current of phase conductor L3 [A]
I leak	Leakage current [A]
P in	Supply unit power consumption [W]
U AC	Emulated line voltage [V]
f AC	Emulated mains frequency [Hz]
I out 24V	Current of the 24 V supply bar

Index

HEIDENHAIN

DR. JOHANNES HEIDENHAIN GmbH
Dr.-Johannes-Heidenhain-Straße 5
83301 Traunreut, Germany
☎ +49 8669 31-0
FAX +49 8669 32-5061
info@heidenhain.de

Technical support   FAX +49 8669 32-1000
Measuring systems ☎ +49 8669 31-3104
  service.ms-support@heidenhain.de
NC support   ☎ +49 8669 31-3101
  service.nc-support@heidenhain.de
NC programming   ☎ +49 8669 31-3103
  service.nc-pgm@heidenhain.de
PLC programming ☎ +49 8669 31-3102
  service.plc@heidenhain.de
APP programming ☎ +49 8669 31-3106
  service.app@heidenhain.de

www.heidenhain.com

Touch probes and vision systems

HEIDENHAIN provides universal, high-precision touch probe systems for machine tools, for example for the exact determination of workpiece edge positions and for tool measurement. Proven technology, such as a wear-free optical sensor, collision protection, or integrated blower/flusher jets for cleaning the measuring point ensure the reliability and safety of the touch probes when measuring workpieces and tools. For even higher process reliability, the tools can be monitored conveniently with the vision systems and tool-breakage sensor from HEIDENHAIN.

For more details on touch probes and vision systems:
www.heidenhain.com/products/touch-probes-and-vision-systems