HEIDENHAIN

10/2023

MANUALplus 620

User's Manual

NC Software
548431-18

Controls and displays

Keys

If you are using a control with touch operation, you can replace some keystrokes with hand gestures.

Further information: "Operating the Touchscreen", Page 101

Keys on visual display unit

Key	Function
	Switch the help graphics between outside and inside machining (only during cycle programming)
	No function
	Select the function in the display unit with the soft key selection key
	Shift between soft-key rows

Operating mode keys

Key	Function
	Select machine operating modes: ■ Machine ■ Teach-in ■ Program run ■ Reference
	Select programming operating modes: ■ smart.Turn ■ DIN PLUS – Unit mode ■ ISO Mode ■ Simulation ■ AWG
	Select tool data and technology data: ■ Tool editor ■ Technology editor
	Select Organization operating mode: ■ Machine parameters ■ Transfer ■ Project management ■ Network connection ■ Diagnostics

Numeric keypad

Key	Function
0 9	Number keys 0 to 9: ▪ Enter numbers ▪ Operate the menu
.	Insert a decimal point
⁻/+	Switch over between positive and negative values
ESC	Escape ▪ Cancel the dialog ▪ Navigate to the top of the menu
INS	Insert ▪ Confirm a dialog ▪ Create a new NC block in the editor
DEL	Delete Delete the selected range
⟨x]	Backspace Erase the character to the left of the cursor
CE	Clear Entry Clear error messages in the Machine operating modes
>>	Enable dialog input fields for further input
ENT	Enter Confirm the entry

Navigation keys

Key	Function
↑ ↓	Move the cursor up or down
← →	Move the cursor to the left or to the right
PG UP PG DN	Page Up and Page Down Scroll back or forward one screen or dialog page
HOME END	Go to beginning of program/list or end of program/list

smart.Turn keys

Key	Function
🗔	Go to the next form
🗔 🗔	Go to next or previous group

Special keys

Key	Function
ERR	Error Open the error window
CALC	Calculator Start the integrated calculator
ℹ	Information ■ Show additional information in the parameter editor ■ Call TURNguide
GOTO	Go to ■ Select alternative input ■ Activate the alphabetic keyboard
DIADUR	DIADUR ■ Use the function in connection with the Remote Desktop Manager ■ Open the HEROS menu

Machine operating panel

Key	Function
	Start or stop machining
	Stop the feed rate
	Stop the spindle
	Switch on the spindle
	Spindle **jog** The spindle rotates as long as you press the key.
	Move the axes (e.g., in +X or +Y direction)
	Change the spindle (machine-dependent)

Operating panel of the control
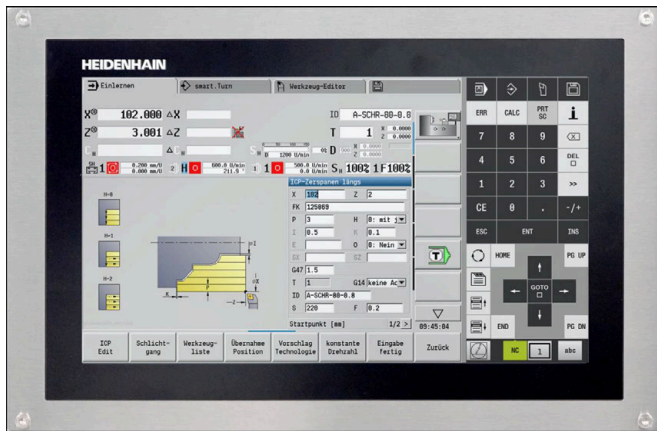
MC 8420T

Table of contents

1

Fundamentals

1.1 About this manual

Safety precautions

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

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

⚠ DANGER

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

⚠ WARNING

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

⚠ CAUTION

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

NOTICE

Notice indicates danger to material or data. If you do not follow the avoidance instructions, the hazard **could result in 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.

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

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

tnc-userdoc@heidenhain.de

1.2 Software and features

This manual describes functions that are available in the control with NC software number 548431-18.

smart.Turn and DIN-PLUS programming are not included in this manual. These functions are described in the User's Manual for smart.Turn and DIN PLUS Programming (ID 685556-xx). Please contact HEIDENHAIN if you require a copy of this manual.

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

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

- Positioning of spindle (**M19**) and driven tool
- Machining with the C or Y axis
- Machining with the B axis

In order to find out about the specific support for your machine, please contact the machine manufacturer.

Many machine manufacturers and HEIDENHAIN offer programming courses. We recommend these courses as an effective way of improving your programming skill and sharing information and ideas with other users of the control.

HEIDENHAIN also offers the DataPilot MP 620 or DataPilot CP 640 software package for PCs, which is designed for use with the respective control. The DataPilot is suitable for both shop-floor programming as well as off-location program creation and testing. It is also ideal for training purposes. The DataPilot can be run on PCs with WINDOWS operating systems. HEIDENHAIN provides the DataPilot as a Windows programming station and running in an Oracle VM Virtual Box. Oracle VM VirtualBox is a software application (virtual machine), in which the control software runs as a separate system, i.e. in a virtual environment.

Intended place of operation

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

Legal information

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

To call further information on the control:

- ▶ Switch to the **Organization** operating mode (floppy disk symbol).
- ▶ Switch to the second level of the soft-key row
- ▶ Press the **LICENSE INFO** soft key

Software options

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

Additional Axis (option 0 to option 7)

Additional axis	Additional control loops 1 to 8

Teach-in (option 8)

Teach-in functionality	■ Contour description with ICP
	■ Cycle programming
	■ Technology database with 9 workpiece-material/tool-material combinations

smart.Turn (option 9)

	■ Contour description with ICP
	■ Programming with smart.Turn
	■ Technology database with 9 workpiece-material/tool-material combinations

Tools and Technology (option 10)

Tool and technology database	■ Tool database expanded to 999 entries
	■ Technology database expanded to 62 workpiece-material/tool-material combinations
	■ Tool life monitoring with exchange tools

Thread Recutting (option 11)

Threads	■ Recutting of threads in **Teach-in** submode
	■ Handwheel superimposition during thread cutting

HEIDENHAIN DNC (option 18)

	Communication with external PC applications over COM component

DXF Import (option 42)

	Loading of DXF contours

B-Axis Machining (option 54)

B-axis machining	■ Tilting the working plane
	■ Tilting the tool carrier
	■ High Dynamic Turning

TURN PLUS (option 63)

	Automatic generation of smart.Turn programs

Parallel Axes (option 94)

Parallel axes	Support of parallel axes (U, V, W)

Spindle Synchronism (option 131)

Spindle synchronism	Synchronism of several lathe spindles

Counter Spindle (option 132)

| **Counter spindle** | ■ Synchronism of main spindle and counter spindle |
| | ■ Rear-face machining |

Remote Desktop Manager (option 133)

| **Remote operation of external comput-er units** | ■ Windows on a separate computer unit |
| | ■ Incorporated in the control's interface |

Synchronizing Functions (option 135)

| **Synchronizing functions** | Advanced synchronization of axes and spindles |

Load Monitoring (option 151)

| **Load monitoring** | Monitoring of the axes and spindles |

New functions of software 548431-18

■ Using **G69**, you define in the NC program which graphics detail the control will display in the simulation or program graphics. This allows you to automatically set the focus of the simulation on the workpiece areas currently being machined. You can additionally delete the previous traverse paths.

■ With **G161**, the control converts traversing movements so that you can use CAM-generated NC programs for machining operations on the counter spindle. The control does not change the NC program, but shows the changes in the basic block display.

■ The HEROS menu has been enhanced:

  ■ In the HEROS settings, you can adjust the screen brightness of the control.

  ■ 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).

  ■ The HEROS tool **Diffuse** has been added. You can compare and merge text files.

Further information: "Overview of taskbar", Page 697

Changed functions of software 548431-18

- You can define the center point or starting point with polar coordinates within the following functions for contour description:
  - **G300 Hole, face**
  - **G401 Pattern linear face**
  - **G402 Pattern linear surf**
  - **G405 DataMatrix face C**
- The cycles **G841** to **G844** for lateral-surface machining with the Y axis and associated units have been enhanced:
  - The parameter **D** allows you to select bidirectional or unidirectional milling direction.
  - If the parameter **D** is defined with the value **1**, you can select between climb milling and up-cut milling by using the parameter **H**.
  - The parameter **FP** allows you to define a special feed rate for the first milling path of each plunging depth. If the first milling path is a full cut, you can use a reduced feed rate to reduce tool wear.

    Only available with **G841** and **G843** cycles.
  - The parameter **J** allows you to select which area of the workpiece the control will machine.

    Only effective with **D=1**.
  - The parameter **O** allows you to select whether the control executes additional infeeds beyond the reference diameter during tilted machining. The additional infeeds reduce tool wear.

    Only effective with **D=1**.
- For ICP figures and patterns for the face, you can define the center point with polar coordinates.

  Further information: "Front face contours in smart.Turn operating mode (option 9)", Page 493

  Further information: "Overview of taskbar", Page 697
- The user administration has been enhanced as follows:
  - Your IT administrator can set up a function user to facilitate connectivity to the Windows domain.
  - If you have connected the control to the Windows domain, you can export the required configurations for other controls.

■ The control uses an icon to indicate whether a connection configuration is secure or insecure.

The control no longer supports connections to PCs running the Windows 7 operating system.

■ The machine parameter **CfgMilling** (no. 602800) with machining parameters for milling has been enhanced:

■ **tmufac** (no. 602809): Overlap factor for trochoidal milling

■ **tmrfp** (no. 602819): Retraction feed rate for trochoidal milling

■ **tmral** (no. 602811): Lift-off distance during retraction for trochoidal milling

■ **tmew** (no. 602812): Angle of the plunge-cut during reciprocating or helical plunge

■ **tmhfac** (no. 602813): Factor by which the plunging helix is greater than the cutter diameter

■ **tmzfac** (no. 602814): Factor by which the infeed feed rate for trochoidal milling deviates from the nominal value

■ **tmefac** (no. 602815): Factor by which the contouring feed rate for trochoidal milling deviates from the nominal value

■ **amufac** (no. 602816): Overlap factor for area milling

■ **amvfac** (no. 602817): Overrun factor for area milling

■ **amzfac** (no. 602818): Factor by which the contouring feed rate for trochoidal milling deviates from the nominal value

Further information: "List of machine parameters", Page 620

2

First Steps

2.1 Overview

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

The following topics are covered in this chapter:

- Switching on the machine
- Setting up tools
- Setting up the workpiece
- Machining the workpiece

> 📖 The following topics are covered in the smart.Turn and ISO Programming User's Manuals:
>
> - Switching on the machine
> - Programming the workpiece
> - Graphically testing the workpiece

2.2   Switching on the machine

> **⚠ 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 on the machine:
▶ Switch on the power supply for the control and the machine
> The control starts the operating system. This process may take several minutes.
> The control displays the **Power interrupted** dialog

CE
    ▶ Press the **CE** key
    > The control compiles the PLC program.
    > The control displays the error message **Switch on external dc voltage**.

I
    ▶ Switch on the machine control voltage
    > The control checks the functioning of the emergency stop circuit.
    > The control is in **Reference** submode.

Z
    ▶ Press the **Z** reference soft key

X
    ▶ Press the **X** reference soft key

All
    ▶ Alternative: Press the **All** soft key

    ▶ Press the **NC start** key
    > The control approaches the reference positions.
    > The control activates the position display and activates the **main menu**.

> ℹ Whether a reference run is required depends on the type of encoders.

Further information on this topic
■ Traversing the reference points
Further information: "Submode Reference", Page 112
■ Operating modes
Further information: "Modes of operation", Page 59

2.3   Setting up tools

Selecting the Tool editor operating mode

To set up tools, select the **Tool editor** operating mode.

▶ Press the **Tool editor** key
> The control switches to the **Tool editor** operating mode.

Further information on this topic

- Tool editor operating mode
  Further information: "Tool editor mode of operation", Page 573
- Tool list
  Further information: "Tool database", Page 570

Preparing and measuring tools

Prepare the tools for machining as follows:

▶ Clamp the required tools in their tool holders

If you measure the tools with an external tool presetter:

▶ Measure the tools
▶ Write down the length and the radius or transfer them directly to the machine using a transfer program
▶ Install the tools

If you measure the tools on the machine:

▶ Install the tool
▶ Measure the tool

Further information on this topic

- Measuring on the machine
  Further information: "Tool measurement", Page 158
- Preparing the tools
  Further information: Refer to your machine manual

Creating new tools

Create a new, externally measured turning tool as follows:

▶ Press the **New tool** soft key

▶ Press the **Turning tool** soft key

> The control opens a dialog window for tool definition.

▶ Enter the input values:

- **ID**: **ID no.** – Tool name (max. 16 characters)
- **TO**: **Tool orientation** (for identification number, see help graphic) (e.g., 1)
- **XL**: **Setup dim. in X** (e.g., 100 mm)
- **ZL**: **Setup dim. in Z** (e.g., 50 mm)
- **YL**: **Setup dim. in Y** (e.g., 0 mm)
- **RS**: **Cutting edge radius** (e.g., 0.8 mm)
- **SL**: **Cutting length** (e.g., 12 mm)
- **EW**: **Setting angle** (e.g., 95°)
- **SW**: **Point angle** (e.g., 55°)
- **MD**: **Turn. direct.** (e.g., 4)
- **QT**: Reference to **Tool text** (e.g., 1) (1 = **Roughing Outside**)

▶ Press the **Save** soft key

> The control adds the tool to the tool list.

Further information on this topic

- Creating new tools
  Further information: "Editing the tool data", Page 575
- Various tool types
  Further information: "Tool types", Page 570
- Tool parameters
  Further information: "General tool parameters", Page 588
- Tool dimensions
  Further information: "Tool dimensions", Page 68

Setting up the turret list

The turret list must be set up before you call a tool. The turret list shows the tools currently in the turret.

Set up the turret list as follows:

▶ Switch to **Machine** operating mode

▶ Select the **Set T, S, F** menu item
> The control opens a dialog window for selecting the tool and the cutting data.

▶ Press the **Turret list** soft key
> The control displays the tools currently in the turret.

▶ Press the **Tool list** soft key
> The control displays the tool list.

▶ Use the **Pocket down** and **Pocket up** soft keys to select the desired line in the turret list

▶ Use the arrow keys to select the desired tool in the tool table

▶ Press the **Load tool** soft key
> The selected tool is loaded into the turret list.
▶ Transfer all required tools to the turret list
▶ Press the **Back** soft key

▶ Press the **Save** soft key
> The control saves the turret assignment list.
▶ Press the **Back** soft key
> The control displays the main menu.

Further information on this topic

■ Set T, S, F menu item
   Further information: "Input of machine data", Page 119
■ Turret list
   Further information: "Pocket table setup", Page 129

2.4 Setting up the workpiece

Clamping the workpiece

Use suitable chucking equipment to mount the workpiece blank safely to the machine.

NOTICE

Danger of collision!

There is a risk of collision between the chucking equipment and the tool. If the free length of the workpiece does not protrude far enough from the chucking equipment, the tool will collide with the chucking equipment!

- Clamp the workpiece such that the free length of the workpiece suffices
- Measure the free length of the workpiece to verify that it is correct
- If required, choose a workpiece with a greater length to ensure a safe setup

---

Defining the tool change position

The control moves the tool to the tool change position every time the tool is changed. In order to ensure safe insertion of the tool, the tool change position needs to be defined. Define the tool change position such that the turret can rotate without collision and the tools do not damage the workpiece during tool change.

Set the tool change position as follows:

▶ Select the **Setup** menu item

▶ Select the **Set tool change point** menu item

▶ Move the tool manually to the desired tool change position

▶ Press the **Take over position** soft key
> The control saves the current position as the tool change position.

▶ Press the **Back** soft key

▶ Press the **Back** soft key
> The control displays the main menu.

Further information on this topic

- Tool change position
  Further information: "Defining the tool change position", Page 144

Inserting the tool

After you have defined the tool change position, you can insert the tool. In the same step, you define the cutting data.

▶ Select the **Set T, S, F** menu item

▶ Enter the desired tool in **T**

▶ Define the desired cutting data:

- **F**: **Feed per revolution** in mm/rev
- **S**: **Cutting speed** in m/min

▶ Press the **Save** soft key

> The selected tool is inserted.

▶ Press the **Back** soft key

> The controls switches to the main menu.

Further information on this topic

- Inserting the tool
  Further information: "Tool call", Page 136
- Defining the cutting data
  Further information: "Input of machine data", Page 119

Defining the workpiece datum

There are several methods for defining the datum on the workpiece. You can set the datum on the plane surface of the workpiece or allow for an oversize when programming the NC program.

Define the workpiece datum as follows:

▶ Select the **Setup** menu item

▶ Select the **Set axis value** menu item

▶ Switch on the spindle

▶ Touch the plane surface of the workpiece manually with the tool

The workpiece datum is not set on the plane surface:

▶ Enter the distance between the tool and the workpiece datum as **Measuring pt. coordinate Z**

> The control calculates the workpiece datum **Z**

▶ Press the **Save** soft key
> The control saves the specified datum.

The workpiece datum is set on the plane surface:
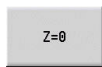
▶ Press the **Z = 0** soft key
> The control saves the current position as the workpiece datum.

▶ Press the **Back** soft key

▶ Press the **Back** soft key
> The control displays the main menu.

Further information on this topic:

■ Setting the workpiece datum
Further information: "Defining the workpiece datum", Page 141

2.5 Machining the workpiece

Teach-in submode (option 8)

Your task is to machine the part shown to the right in **Teach-in** submode. To do this, you program the machining operations using teach-in cycles. The control saves the programmed cycles to an NC program.

Opening a new NC program

Open a new cycle program as follows:

▶ Switch to **Machine** operating mode

▶ Press the **Teach-in** soft key

▶ Press the **Program list** soft key
▶ Enter the file name
▶ Press the **Open** soft key
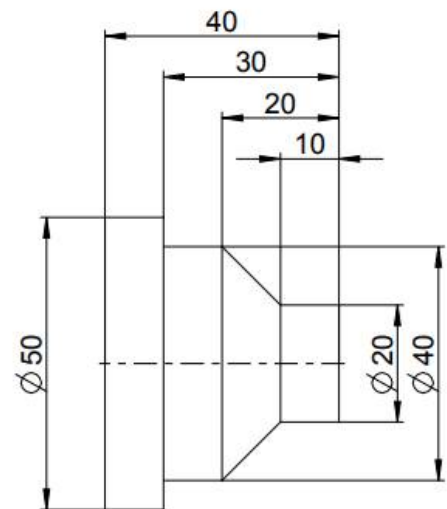
Defining the workpiece blank

▶ Press the **Add cycle** soft key

▶ Select the **Define workpiece blank** menu item

▶ Select the **Bar/tube blank** menu item
> The control opens a dialog window.
▶ Define the parameters:
  - **X**: **Outside diameter** = 60 mm
  - **Z**: **Length** – Including transverse oversize and clamping range = 60 mm
  - **K**: **Righthnd edge** – Transverse oversize = 1 mm
  - **RG**: **Activate contour follow-up** = 1: With blank form update

▶ Press the **Input finished** soft key

▶ Press the **Simulation Start** soft key
> The control switches to **Simulation** submode.

▶ Press the **Save** soft key
> The control switches to **Teach-in** submode.

Transverse turning of the workpiece

Add cycle	▶ Press the **Add cycle** soft key
	▶ Select the **Single cuts** menu item
	▶ Select the **Linear mach. transv.** menu item
	> The control opens a dialog window.
With return	▶ Press the **With return** soft key
	▶ Define the parameters:

- **X**: **Start point** = 62 mm
- **Z**: **Start point** = 2 mm
- **Z1**: **Start point contour** (if **With return** is active) = 0 mm
- **X2**: **End point contour** − Twice the radius of the turning tool (e.g., −1.6 mm)
- **T**: **Tool number** − turret pocket number
- **S**: **Cutting speed** or **Shaft speed constant** (e.g., 220 m/min)
- **F**: **Feed per revolution** (e.g., 0.2 mm/rev)

Input finished	▶ Press the **Input finished** soft key
	▶ Press the **Simulation Start** soft key
	> The control switches to **Simulation** submode.
	> The control simulates the machining cycle.
Save	▶ Press the **Save** soft key
	> The control switches to **Teach-in** submode.

> After you have defined a cycle, you can immediately run it.

Machining the workpiece contour

▶ Press the **Add cycle** soft key

▶ Select the **Roughing cycles lon/trans** menu item

▶ Select the **ICP cut longitud.** menu item

▶ Press the **Edit ICP** soft key
> The control opens the **ICP contours**.
▶ Enter the name of the contour
▶ Press the **Open** soft key
> The control switches to **ICP editor** submode.
▶ Select the **Contour** menu item

▶ Select the **Line** menu item
▶ Enter the coordinates:
  - **XS**: **Start point** of contour = 0 mm
  - **ZS**: **Starting point** of contour = 0 mm
  - **X**: **Target point** = 20 mm
▶ Press the **Save** soft key

▶ Select the **Line** menu item
▶ **Z**: **Target point** = −10 mm
▶ Press the **Save** soft key

▶ Select the **Line** menu item
▶ Enter the coordinates:
  - **X**: **Target point** = 40 mm
  - **Z**: **Target point** = −20 mm
▶ Press the **Save** soft key

▶ Select the **Line** menu item
▶ **Z**: **Target point** = −30 mm
▶ Press the **Save** soft key

▶ Select the **Line** menu item
▶ **X**: **Target point** = 50 mm
▶ Press the **Save** soft key

▶ Select the **Line** menu item
▶ **Z**: **Target point** = −40 mm
▶ Press the **Save** soft key

▶ Select the **Line** menu item
▶ **X**: **Target point** = 60 mm

Save	▶ Press the **Save** soft key
Back	▶ Press the **Back** soft key
Back	▶ Press the **Back** soft key > The control switches to **Teach-in** submode. ▶ Define the parameters: ■ **X**: **Start point** = 65 mm ■ **Z**: **Start point** = 2 mm ■ **P**: **Feed depth** – Maximum infeed depth (e.g., 5 mm) ■ **T**: **Tool number** – turret pocket number ■ **S**: **Cutting speed** or **Shaft speed constant**, e.g. 220 m/min ■ **F**: **Feed per revolution**, e.g. 0.35 mm/rev
Input finished	▶ Press the **Input finished** soft key
	▶ Press the **Simulation Start** soft key > The control switches to **Simulation** submode. > The control simulates the machining cycle.
Save	▶ Press the **Save** soft key > The control switches to **Teach-in** submode.
Back	▶ Press the **Back** soft key > The control displays the main menu.

Further information on this topic

■ Teach-in submode
Further information: "Working with cycles", Page 196
■ Creating an ICP contour
Further information: "Programming ICP contours", Page 444
■ Simulation submode
Further information: "Graphic Simulation", Page 543

Program run submode

In **Program run** submode, you can select and run NC programs.

By default, the control displays the most recently used NC program.

Load an NC program as follows:

Program run	▶	Press the **Program run** soft key
	>	The control opens the **Program run** submode.
	>	The control displays the most recently used NC program.
Program list	▶	Press the **Program list** soft key
	>	The control displays a dialog window with teach-in programs.
DIN	▶	Press the **ISO** soft key if required
	>	The control displays NC programs from the **smart.Turn** operating mode.
	▶	Select the desired NC program
Open	▶	Press the **Open** soft key
	>	The control loads the NC program.

Start the program run:

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

You can also run an NC program in Single Block mode (when you run a new NC program for the first time, for example). In this mode, the control stops after each traverse path (basic block).

Start the program run in Single Block mode as follows:

Single block	▶	Press the **Single Block** soft key
Base blocks	▶	If required, press the **Base blocks** soft key
	>	The control displays the individual traverse paths during program run.
⬚	▶	Press the **NC start** key for each traverse path.
	>	The control machines the single block.

Further information on this topic

- Running NC programs
  Further information: "Program run submode", Page 168
- Programming in Teach-in mode
  Further information: "Working with cycles", Page 196
- Programming in smart.Turn mode
  Further information: smart.Turn and DIN Programming User's Manual

3

Introduction and Fundamentals

3.1 Fundamentals on the MANUALplus 620 control

The control was conceived for CNC lathes. It is suitable for horizontal and vertical lathes. The control supports machines with a tool magazine or tool turret. The tool carrier of horizontal lathes can be located in front of or behind the workpiece.

The control supports lathes with spindle, one slide (X and Z axis), C axis or positionable spindle, driven tool, and machines with a Y axis.

The MANUALplus for cycle lathes

On the MANUALplus 620, rework or simple tasks can be done in the same way as on a conventional lathe. You move the axes in the normal manner by turning the handwheels. The cycles of the MANUALplus 620 are available for difficult cuts like tapers, undercuts or threads. For small and medium-size production runs, cycle programming will increase your revenue. When machining the first workpiece, you can store the machining cycles, and already save valuable time when machining the second workpiece. For increased requirements and complex machining tasks, you will benefit from the **smart.Turn** programming mode.

MANUALplus for CNC lathes

With the MANUALplus 620, you can position up to four axes, the traverse paths will be interpolated.

Regardless of whether you are turning simple parts or complex workpieces, the MANUALplus 620 provides you with the benefits of graphical contour input and convenient programming with the **smart.Turn** operating mode. Programming with variables, controlling special machine components, or using externally created programs is no problem: Simply switch to DIN PLUS. This programming mode helps you solve all your special tasks. The MANUALplus 620 supports operations with the C axis in cycle, smart.Turn, and ISO programming. In the Y axis, the MANUALplus 620 supports operations with smart.Turn and ISO programming.

3.2 Configuration

In the standard version, the control is equipped with the axes X and Z and a main spindle. Optionally, a C axis, a Y axis, and a driven tool can be configured.

Slide position

The machine tool builder configures the control depending on the slide position.

- Z axis **horizontal** with tool slide behind the workpiece
- Z axis **horizontal** with tool slide in front of the workpiece
- Z axis **vertical** with tool slide to the right of the workpiece

The menu symbols, help graphics and graphic representations during ICP and simulation consider the slide position.

The representations in this User's Manual assume a lathe with a tool carrier centered behind the workpiece.

Tool carrier systems

The control supports the following systems as tool carriers:

- Multifix holder with **one** tool mount
- Turret with **n** tool mounts
- Turret with **n** tool mounts and **one** multifix holder with one tool mount. Here it is possible for one of the two tool carriers to be arranged mirrored on the workpiece side opposite from the standard tool carrier
- Two multifix holders, each with **one** tool mount. The tool carriers are arranged opposite to each other. One of them is then mirrored.
- Magazine with **n** tool mounts and one tool carrier in the working space with one tool mount

C axis (option 55)

With a C axis you can drill and mill a workpiece on its face and lateral surfaces.

When the C axis is used, one axis interpolates linearly or circularly with the spindle in the given working plane, while the third axis interpolates linearly.

The control supports part program creation with the C axis in:

- **Teach-in** submode
- **smart.Turn** operating mode (option 9)
- DIN PLUS programming

Y axis (option 70)

With a Y axis you can drill and mill a workpiece on its face and lateral surfaces.

During use of the Y axis, two axes interpolate linearly or circularly in the given working plane, while the third axis interpolates linearly. This enables you to machine slots or pockets, for example, with plane floors and perpendicular edges. By defining the spindle angle, you can determine the position of the milling contour on the workpiece.

The control supports part program creation with the Y axis in:

- **Teach-in** submode
- **smart.Turn** operating mode (option 9)
- DIN PLUS programming
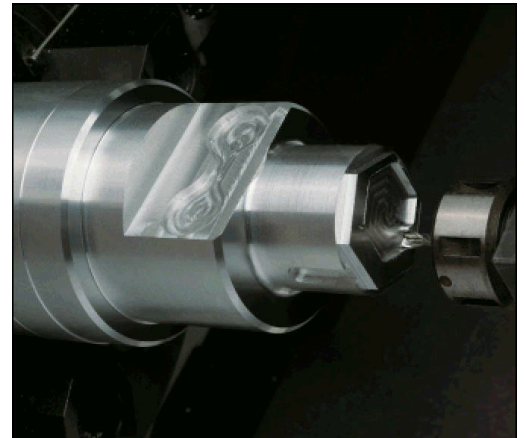
Full-surface machining

Functions that ensure perfectly timed machining and easy programming for multi-operation machining include the following:

- Angle-synchronous part transfer with spindle in rotation
- Traversing to a fixed stop
- Controlled parting
- Coordinate transformations

The control supports full-surface machining for all common machine designs with:

- Rotating gripping device
- Movable counter spindle
- Several spindles and tool carriers
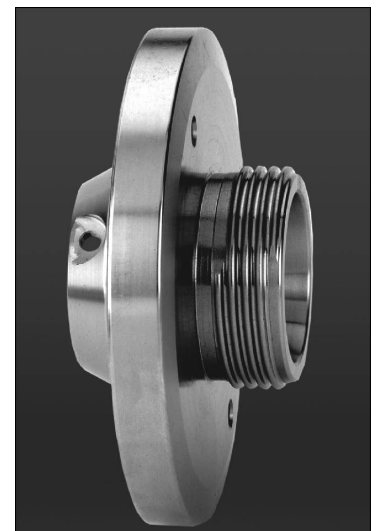
3.3 Features

Configuration

- Basic version: X and Z axis, spindle
- Positionable spindle and driven tool
- C axis and driven tool
- Y axis and driven tool
- B axis for machining in a tilted plane
- Digital current and speed control

Modes of operation

Machine operating mode

Manual slide movement through axis-direction keys or electronic handwheels.

Graphic support for entering and running Teach-in cycles without saving the machining steps in alternation with manual machine operation.

Thread reworking (thread repair in a second workpiece setup).

Teach-in submode

Sequential linking of Teach-in cycles, where each cycle is run immediately after input, or is graphically simulated and subsequently saved.

Program run submode

All are possible in single-block and full-sequence modes:

- DIN PLUS programs
- smart.Turn programs
- Teach-in programs

Setup functions in Machine operating mode

- Setting the workpiece datum
- Defining the tool change point
- Defining the protection zone
- Tool measurement through touch-off, touch probe or optical gauge

Programming

- Teach-in programming
- Interactive contour programming (**ICP**)
- smart.Turn programming
- Automatic program creation with **TURN PLUS**.
- DIN PLUS programming

Simulation

- Graphic representation of the sequence of smart.Turn or DIN PLUS programs and graphic representation of a teach-in cycle or a teach-in program
- Simulation of the tool paths as wire-frame or cutting-path graphics, special identification of the rapid-traverse paths
- Motion simulation (2-D material-removal graphic)
- Side or face view, or 2-D view of cylindrical surface
- Display of programmed contours
- Shifting and magnifying functions

Tool system

- Database for 250 tools
- Database for 999 tools (with option 10)
- Description can be entered for every tool
- Optional support of multipoint tools (tools with multiple reference points or multiple cutting edges)
- Turret or multifix system
- Optional tool magazine

Technology database

- Cutting data is entered in the cycle or in the UNIT as default values
- 9 workpiece-material/tool-material combinations (144 entries)
- 62 workpiece-material/tool-material combinations (992 entries) (with option 10)

Interpolation

- Straight line: in 2 principal axes (max. ±100 m)
- Circular: in 2 axes (radius max. 999 m)
- C axis: Interpolation in the linear axes X and Z with the C axis
- Y axis: Linear or circular interpolation of two axes in the given plane. The respective third axis can simultaneously perform linear interpolation.
    - **G17**: XY plane
    - **G18**: XZ plane
    - **G19**: YZ plane
- B axis: Drilling, boring and milling operations in oblique planes

3.4   Data backup

HEIDENHAIN recommends saving new programs and files created on a PC at regular intervals.

HEIDENHAIN provides a backup function for this purpose in the data transfer software TNCremo. Your machine tool builder can provide you with a copy. You additionally need a data medium on which all machine-specific data, such as the PLC program, machine parameters, etc., are stored.

Please contact your machine tool builder for this.

3.5 Explanation of terms

- **Cursor**: **Marks** the current position in a list or input field

  Input and operations, such as copying, deleting, inserting, etc., refer to the current cursor position.

- **Cursor keys**: Keys for moving the cursor
  - **Arrow keys**
  - **PG UP** and **PG DN** keys

- **Active window, functions, or menu items**: Item on the screen shown in color.

  In the inactive windows, the title bar appears **dimmed**. Inactive function keys or menu items also appear **dimmed**

- **Menu**: Functions or function groups are displayed as a 9-field box

- **Menu item**: Individual symbols of a menu

- **Default value**: Preset value of a cycle parameter or DIN command parameter

- **File name extension**: String following the file name

  Example:
  - ***.nc**: DIN programs
  - ***.ncs**: DIN subprograms (DIN macros)

- **Soft key**: Functions available at the side and bottom of the screen pages

- **Soft key selection keys**: Keys that you can press to select a soft key function.

- **Form**: Individual pages of a dialog

- **UNITS**: Grouped dialogs of a function in **smart.Turn** mode of operation.

3.6 Control design

The dialog between machinist and control takes place via:

- Screen
- Soft keys
- Keyboard
- Machine operating panel

The entered data can be displayed and checked on the screen. With the soft keys directly below the screen, you can select functions, capture position values, confirm entries, and a lot more.

With the **ERR** key you can call error and PLC information.

The data input keyboard (on the operating panel) is used to enter machine data, position data, etc. For the MANUALplus 620, an alphanumeric keyboard is not necessary. When you enter tool descriptions, program descriptions or comments in an NC program, an alphabetic keyboard is displayed on the screen. The machine operating panel contains all necessary controls for manual operation of the lathe.

Cycle programs, ICP contours, and NC programs are stored in the internal memory of the control.

For data exchange and data backup, you can use the **Ethernet interface** or the **USB interface**.

> ℹ️ If you are using a control with touch operation, you can replace some keystrokes with hand gestures.
>
> **Further information:** "Operating the Touchscreen", Page 101

3.7 Fundamentals

Position encoders and reference marks

The machine axes are equipped with position encoders that register the positions of the slide or tool. When a machine axis moves, the corresponding position encoder generates an electrical signal. The control evaluates this signal and calculates the precise actual position of the machine axis.

If there is a power interruption, the calculated position will no longer correspond to the actual position of the machine slide. To recover this assignment, incremental position encoders are provided with reference marks. The scales of the position encoders contain one or more reference marks that transmit a signal to the control when they are crossed over. This enables the control to re-establish the assignment of the displayed position to the current machine position. For linear encoders with distance-coded reference marks, you only need to move each axis a maximum of 20 mm (0.8 in.), and a maximum of 20° for angle encoders.

If incremental position encoders are without reference marks, fixed reference positions have to be traversed after switch-on. The control knows the exact distance between these reference positions and the machine datum (see figure).

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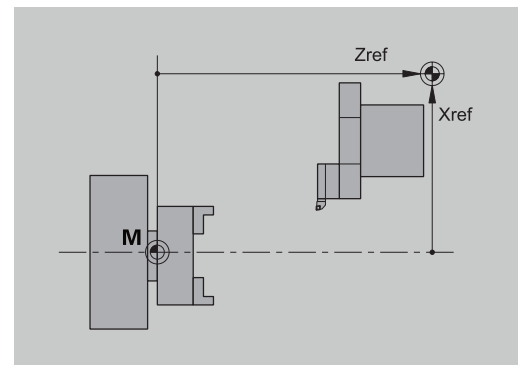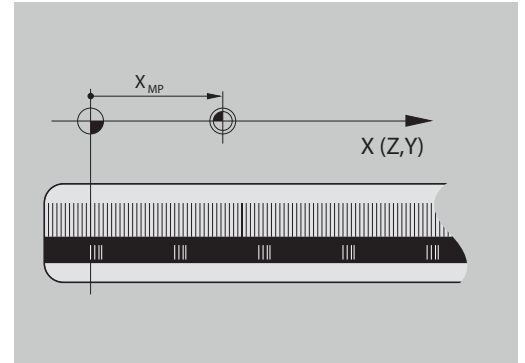
Axis designations

The cross slide is referred to as the **X axis** and the saddle as the **Z axis**.

All X-axis values that are displayed or entered are regarded as **diameters**.

Lathes with **Y axis**: The Y axis is perpendicular to the X axis and Z axis (Cartesian system).

When programming paths of traverse, remember to:

- Program a **positive value** to depart the workpiece
- Program a **negative value** to approach the workpiece
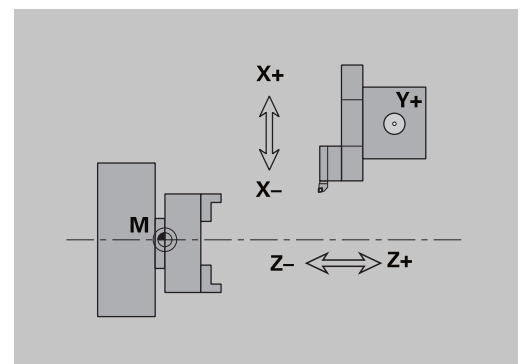
Coordinate system

The meanings of the coordinates X, Y, Z, and C are specified in DIN 66 217.

The coordinates entered for the principal axes X, Y and Z are referenced to the workpiece datum. The angles entered for the rotary axis (C axis) are referenced to the datum of the C axis.

The axis designations X and Z describe positions in a two-dimensional coordinate system. As you can see from the figure to the center right, the position of the tool tip is clearly defined by its X and Z coordinates.

The control can connect points by linear and circular traverses (interpolations). Workpiece machining can be programmed by entering the coordinates for a succession of points and connecting the points by linear or circular traverses.

Like the paths of traverse, you can also describe the complete contour of a workpiece by defining single points through their coordinates and connecting them by linear or circular paths of traverse.

Positions can be programmed to an accuracy of 1 µm (0.001 mm). This is also the accuracy with which they are displayed.
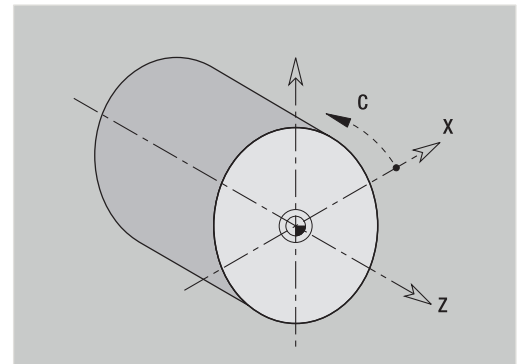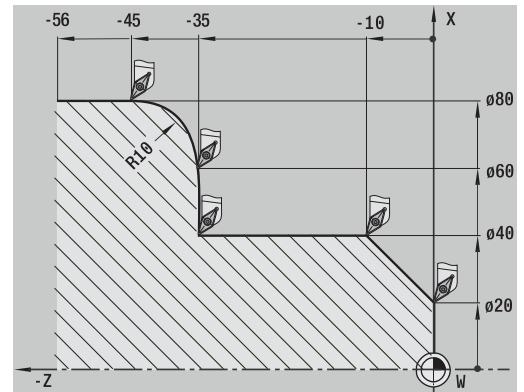
Absolute coordinates

If the coordinates of a position are referenced to the workpiece datum, they are referred to as absolute coordinates. Each position on a workpiece is clearly defined by its absolute coordinates.
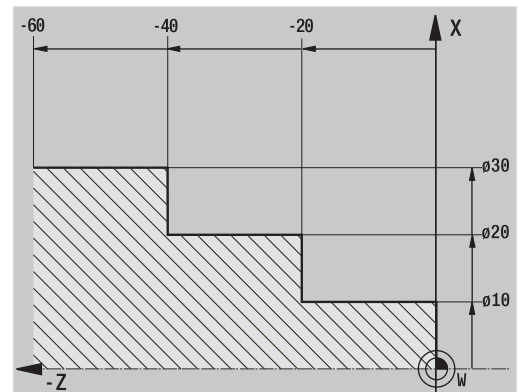
Incremental coordinates

Incremental coordinates are always given with respect to the last programmed position. They specify the distance from the last active position to the subsequent position. Each position on a workpiece is clearly defined by its incremental coordinates.
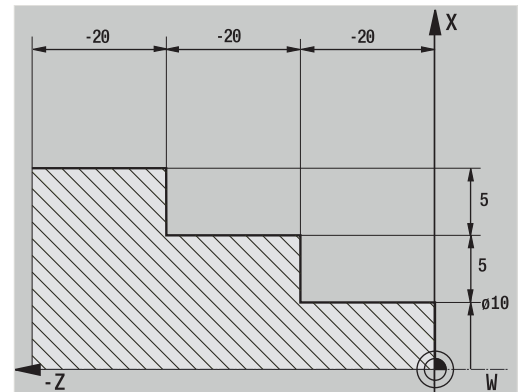
Polar coordinates

Positions located on the face or lateral surface can either be entered in Cartesian coordinates or polar coordinates.

When programming with polar coordinates, a position on the workpiece is clearly defined by the entries for diameter and angle.
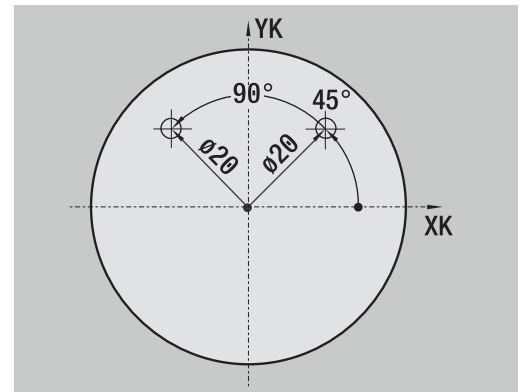
Machine datum

The point of intersection of the X and Z axes is called the **machine datum**. On a lathe, the machine datum is usually the point of intersection of the spindle axis and the spindle surface. The machine datum is designated with the letter **M**.
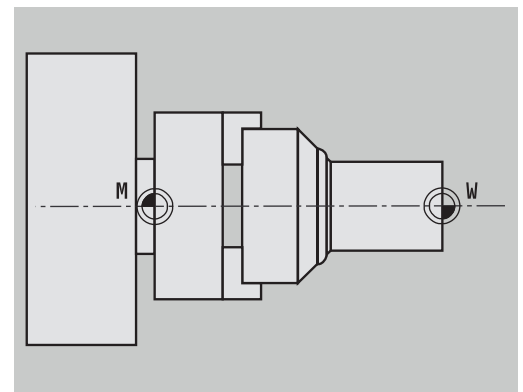
Workpiece datum

To machine a workpiece, it is easier to enter all input data with respect to a zero point located on the workpiece. By programming the zero point used in the workpiece drawing, you can take the dimensions directly from the drawing, without further calculation (dimensioning origin). This point is the workpiece datum. The workpiece datum is designated with the letter **W**.
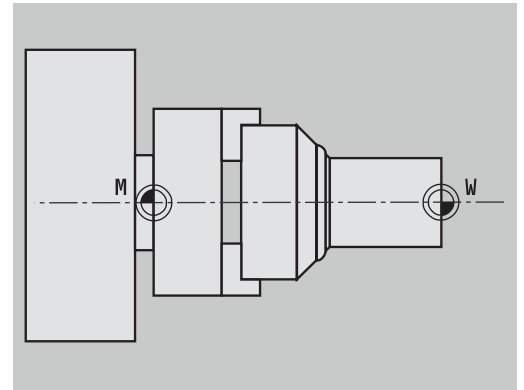
Units of measure

You can program the control either in the **metric** or **inch** system. The units of measurement listed in the table below apply to all inputs and displays.

Dimensions	Metric	Inches
Coordinates	mm	inch
Lengths	mm	inch
Angle	Degrees	Degrees
Spindle speed	rpm	rpm
Cutting speed	m/min	ft/min
Feed per revolution	mm/rev	inch/rev
Feed per minute	mm/min	inch/min
Acceleration	m/s^2	ft/s^2

3.8 Tool dimensions

The control requires information on the specific tools for a variety of tasks, such as positioning the axes, calculating the cutting radius compensation or the proportioning of cuts.

Tool length

All programmed and displayed position values are given with respect to the distance between the tool tip and workpiece datum. Since the control only knows the absolute position of the tool carrier (slide), it needs the dimensions **XL** and **ZL** to calculate and display the position of the tool tip.
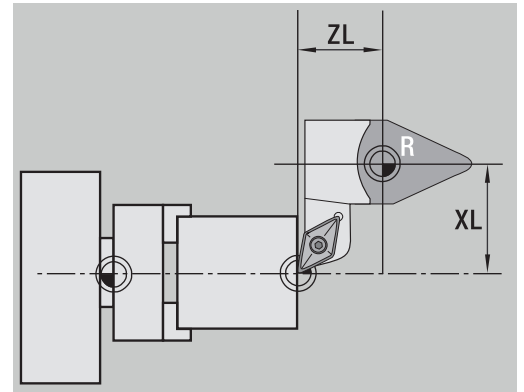
Tool compensation

The tool tip is subjected to wear during machining processes. To compensate for this wear, the control uses compensation values. The compensation values are managed independent of the values for length. The system automatically adds the compensation values to the values for length.

Tool-tip radius compensation (TRC)

The tip of a lathe tool has a certain radius. When machining tapers, chamfers and radii, this results in inaccuracies which the control compensates with its cutting radius compensation function.

Programmed paths of traverse are referenced to the theoretical tool tip **S**. With non-paraxial contours, this will lead to inaccuracies during machining.

The TRC function compensates for this error by calculating a new path of traverse, the **equidistant line**.

The control calculates the TRC for cycle programming. The smart.Turn and DIN programming feature also takes the TRC into account in clearance cycles. During DIN programming with single paths, you can also enable/disable TRC.

The control displays a warning if residual material is left behind due to the angle of the cutting edges or the tool angle. You can suppress the warning with machine parameter **suppressResMatlWar** (no. 201000).
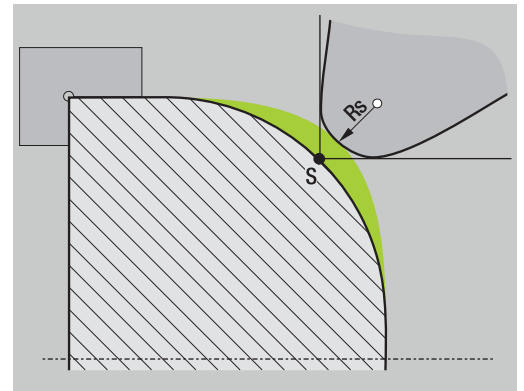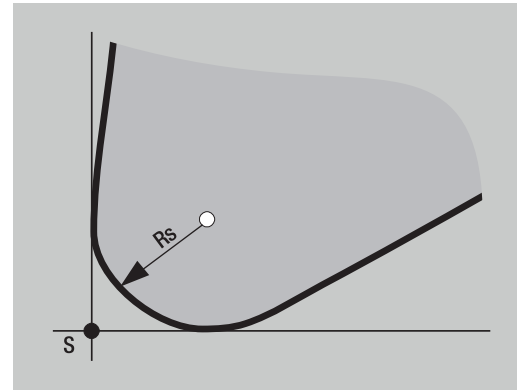
Milling cutter radius compensation (MCRC)

In milling operations, the outside diameter of the milling cutter determines the contour. When the MCRC function is not active, the center of the cutter is used as the point of reference. The MCRC function compensates for this error by calculating a new path of traverse, the **equidistant line**.
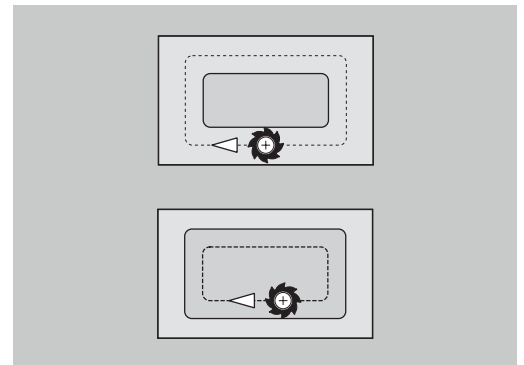
4

Basics of Operation

4.1   General information on operation

Operation

- Select the desired operating mode with the corresponding operating mode key
- Within the operating mode, you can change the mode through the soft keys
- With the numeric keypad you can select the function within the menus
- Dialogs can consist of multiple pages
- Besides with the soft keys, dialogs can be concluded positively with **INS** or negatively with **ESC**
- Changes made in lists are effective immediately

  These changes are retained even if the list is closed using **ESC** or **CANCEL**.

Setup

- You will find all setup functions in the **Machine** operating mode in **Manual mode**
- All preparatory work can be performed using the **Setup** and **Set T, S, F** menu items

Program name

The **program name** starts with a digit or a letter, followed by up to 40 characters and the **.nc** extension for main programs and the **.ncs** extension for subprograms.

For program names, you can use any ASCII characters except:

~ * ? < > | / \ : " % #

The following characters have special meanings:

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

Programming in Teach-in operating mode

▶ Select the **Machine** operating mode

▶ Select the **Teach-in** submode

▶ Press the **Program list** soft key

▶ Open a new cycle program

▶ Press the **Add cycle** soft key to enable the cycle menu

▶ Select and specify the desired machining operations

▶ Press the **Input finished** soft key

▶ Start the simulation and check the machining sequence

▶ Select the desired graphics options, if applicable

▶ Press **NC Start** to start machining

▶ Save the cycle after the operation is completed

▶ Repeat the above steps for each new machining operation

Programming in smart.Turn operating mode (option 9)

- Convenient programming with **Units»** in a structured NC program
- Combinable with DIN functions
- Contour definition graphically possible
- Contour follow-up when used with a workpiece blank
- Conversion of cycle programs to smart.Turn programs with the same functions

4.2   Control screen

The control shows the data to be displayed in windows. Some windows only appear when they are needed, for example, for typing in entries.

In addition, the control shows the **type of operation**, the **soft-key display** and the **PLC soft key display** on the screen. Each function that appears in a field of the soft-key row is activated by pressing the soft key directly below it.

> **ⓘ**   If you are using a control with touch operation, you can replace some keystrokes with hand gestures.
>
> **Further information:** "Operating the Touchscreen", Page 101

Operating mode line

The operating mode tabs (at the top of the screen) show the four operating modes as well as the submodes.

Machine display

The machine display field (beneath the operating mode tabs) is configurable. It shows all important information on axis positions, feed rates, rotational speeds, and tools.

Other windows used

- **List and program windows**: Here, program, tool, and parameter lists, etc. are displayed

  You can use the arrow keys to **navigate** within the list and select the list items to be edited.

- **Menu window**: Displays the menu symbols

  This window only appears on the screen in **Teach-in** submode and **Machine** operating mode

- **Input window / dialog window**: For entering the parameters of a cycle, ICP element, DIN command, etc.

  View the existing data, then delete or edit them in the dialog window.

- **Help graphics**: Input data (such as cycle parameters, tool data, etc.) are explained with graphics.

  The **triple-arrow key** (at the left edge of the screen) allows you to switch between the help graphics for inside and outside machining (only for cycle programming).

- **Simulation window**: The simulation window shows a graphic representation of the contour elements and a simulation of the tool movements.

  This way, you can check cycles, entire cycle programs, and DIN programs.

- **ICP contour graphics**: Display of the contour during ICP programming

- **DIN editing window**: Display of the DIN program during DIN programming

- **Error window**: Display of encountered errors and warnings
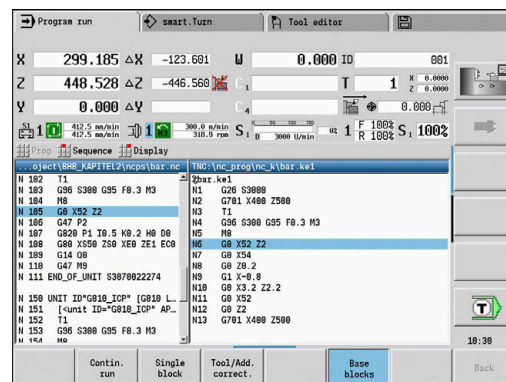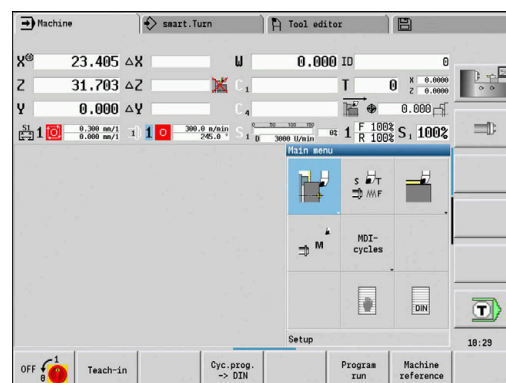
4.3 Operation and data input

Modes of operation

The active mode of operation is highlighted in the operating-mode tab. The control differentiates between the following operating modes:

- **Machine**—with the submodes:
    - **Teach-in**
    - **Program run**
    - **ICP editor**
    - **Reference**
    - **Simulation**
- **smart.Turn**—with the submodes:
    - **ICP editor**
    - **Automatic working plan generation AWG**
    - **Simulation**
- **Tool editor**—with the submodes:
    - **Technology editor**
- **Organization**—with the submodes:
    - **Machine par. programming**
    - **Transfer**

You can use the operating mode keys to switch between the modes. The selected submode and the current menu position remain when the operating mode is switched.

If you press the operating mode key in a submode, the control switches back to the main level of the mode.
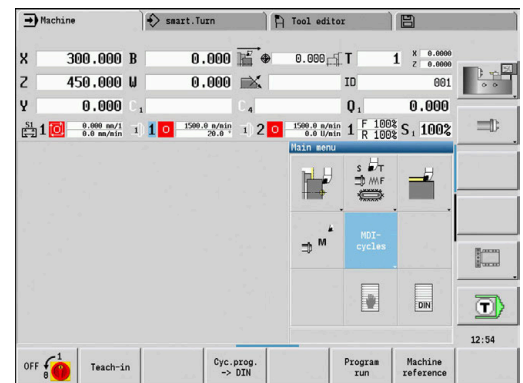
> **ℹ** In some situations, it is not possible to switch operating modes (e.g., while a tool is being edited in **Tool editor** operating mode).
>
> In such cases, you need to terminate the editing process or close the dialog before switching the operating mode.

Menu selection

The numerical keypad is used for activating a menu and for entering data. They are displayed differently depending on the operating mode:

- During setup, for example in **Teach-in** submode, the functions are shown in a 9-field box, the **menu window**

  The meaning of the selected symbol / menu item is described in the bottom line.

- In other operating modes, the keypad symbol is shown with the position of the function marked.

Press the corresponding numerical key, or move the highlight with the arrow keys to the symbol on the screen and press the **ENT** key.
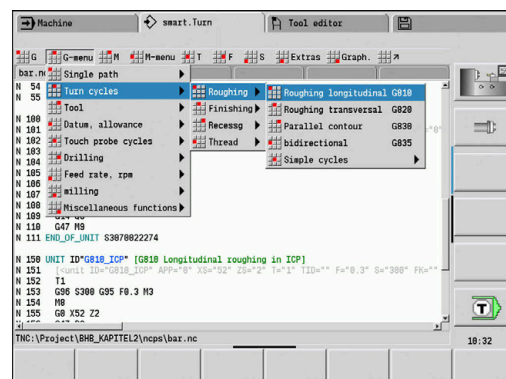
Soft keys

- With some system functions, the available functions are arranged on several soft-key levels

- Some soft keys work like **toggle switches**.

  A function is active when the associated field in the soft-key row is **highlighted** in color. The setting remains in effect until the function is switched off again.

- With functions like **Take over position**, you do not have to enter values manually

  The data are automatically written into the appropriate input fields.

- Data input is not concluded until you press the **Save** or **Input finished** soft key

- The **Back** soft key takes you back to the previous operating level

Data input

Input windows comprise several **input fields**. You can move the cursor to the desired input field with the **up** or **down** arrow key. The control shows the function of the selected field in the bottom line of the window or directly before the input field.

Place the highlight on the desired input field and enter the data. Existing data are overwritten. With the **left** or **right** arrow key, you can move the cursor **within** the input field and place it on the position where you want to delete, copy or add characters.

To confirm the data you entered in a field, press the **up** or **down** arrow key or the **ENT** key.

If there are more input fields than a window can show, a second input window is used. You will recognize this through the symbol in the bottom line of the input window. To switch back and forth between the windows, press the **PgUp/PgDn** keys.

> ℹ️ Data entry is concluded when you press the **OK**, **Input finished**, or **Save** soft key. The **Back** or **Cancel** soft key discards input or changes.

smart.Turn dialogs

The unit dialog is divided into fillable forms and the forms are divided again into groups. The forms are identified by tabs and fine lines divide each tab into groups. You can navigate between the forms and groups with the smart.Turn keys.

smart.Turn keys

🖹		Go to the next form
🗐▾	🗐▴	Go to next or previous group

```
G820 Roughing, direct transverse
Overvw.   Tool   Contour   Cycle   Global ◄►

Appro… XS 52          Appro… ZS 2
ID number           TID 5
⋯▸ Feed                 F 0.3
◉ Cutting speed         S 300
Start point contour     X1 50
Start point contour     Z1 0
End point contour       X2 0
End point contour       Z2 1
Maximum infeed          P 2
Oversize X              I 0.500
Oversize Z              K 0.200

Approach position X [mm]        1/7
```

List operations

The control displays cycle programs, DIN programs, tool lists, etc. as lists. Use the arrow keys to navigate within a list if you want to view data or select elements for deletion, copying, editing, etc.

Alphabetic keyboard

You enter letters and special characters with the screen keypad or (if available) with a PC keyboard connected over the USB port.

Cleaning

Switch the control off before cleaning the keyboard unit.

NOTICE
Caution: danger 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 cleaning agents 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

> **i** 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 when it stops functioning 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

> **i** 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

Entering text with the screen keyboard

▶ Press the **Alphabetic keyboard** soft key or the **GOTO** key to enter text.

> The control opens the **Text input** window.

▶ Enter the desired character or special character by pressing the numerical key multiple times.

▶ Use the **abc/ABC** soft key to select upper or lower case.

▶ Wait until the selected character appears in the input field.

▶ Then, enter the next character.

▶ Use the **OK** soft key to transfer the text into the open dialog field. To delete individual characters, use the **BACKSPACE** soft key.
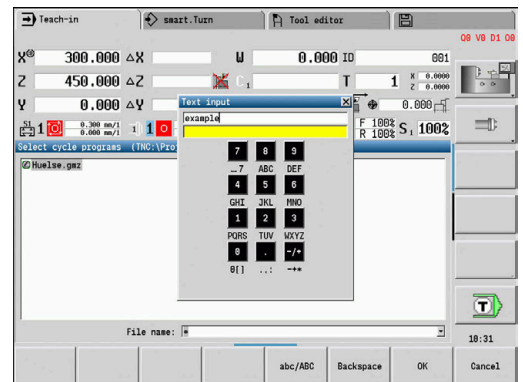
4.4 Calculator

Calculator functions

The calculator can be selected only from open dialogs in cycle programming or smart.Turn programming.

You can use the calculator in the following three **views**:

- Scientific
- Standard
- Equation editor: Here, you can perform multiple calculations in immediate sequence (example: 17*3+5/9)

> ℹ The calculator remains active even after switching the operating mode. Press the **END** soft key to close the calculator.
>
> The **GET VALUE** soft key enables you to transfer a numerical value from the active input field to the calculator. The **CONFIRM** soft key enables you to transfer the current value from the calculator to the active input field.
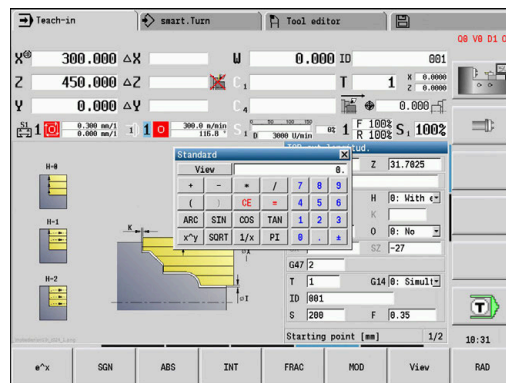
Using the calculator

| ↓ | ▶ Use the cursor keys to select the input field. |

| CALC | ▶ Use the **CALC** key to activate or deactivate the calculator |

| ▷ | ▶ Shift the soft-key menu until the desired function appears |

To perform the calculation:

| CONFIRM VALUE | ▶ Press the **CONFIRM** soft key |
| | ⟩ The control transfers the value into the active input field and closes the calculator. |

To switch the view of the calculator:

| View | ▶ Press the **View** soft key until the desired view is set |
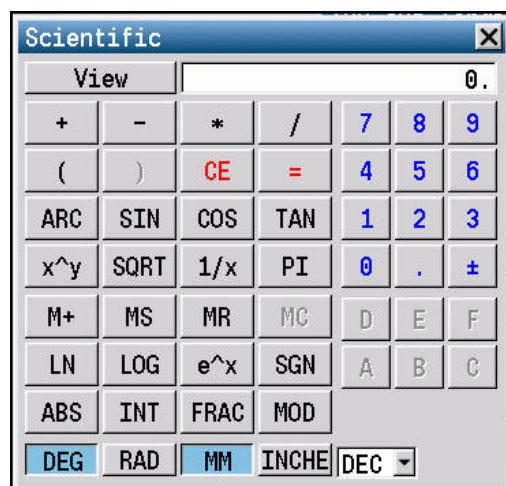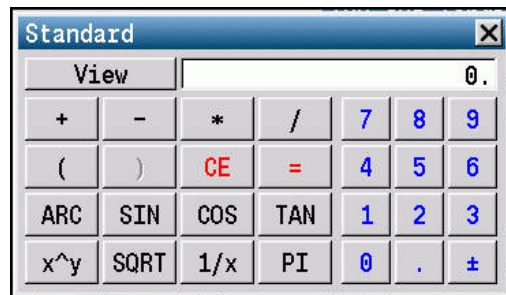
Calculate function	Shortcut (soft key)
Addition	**+**
Subtraction	**-**
Multiplication	*****
Division	**/**
Calculating with parentheses	**()**
Arc	**ARC**
Sine	**SIN**
Cosine	**COS**
Tangent	**TAN**

Calculate function	Shortcut (soft key)
Powers of values	**x^y**
Square root	**SQRT**
Inversion	**1/x**
pi (3.14159265359)	**PI**
Add value to buffer memory	**M+**
Save the value to buffer memory	**MS**
Recall from buffer memory	**MR**
Delete buffer memory contents	**MC**
Natural logarithm	**LN**
Logarithm	**LOG**
Exponent function	**e^x**
Check the algebraic sign	**SGN**
Form the absolute value	**ABS**
Truncate decimal places	**INT**
Truncate places before the decimal point	**FRAC**
Modulus operator	**MOD**
Select view	**View**
Delete value	**DEL**
Unit of measure	**MM** or **INCH**
Display mode for angle values	**DEG** (degree) or **RAD** (radian measure)
Display mode of the numerical value	**DEC** (decimal) or **HEX** (hexadecimal)

The arc calculate function only has a function in connection with **SIN**, **COS** or **TAN**.

The calculator writes the inverse function as **ASIN**, **ACOS** or **ATAN**.

Adjusting the position of the calculator

To move the calculator:

▶ Move the calculator with the arrow keys

You can also move the calculator with a connected mouse.
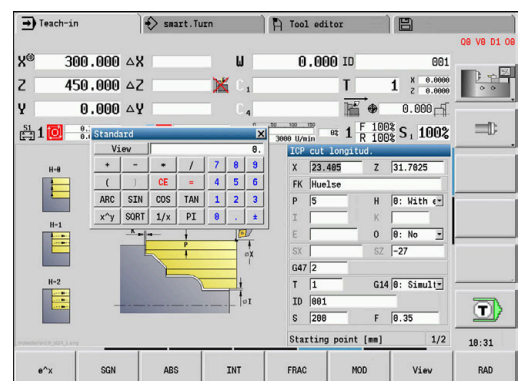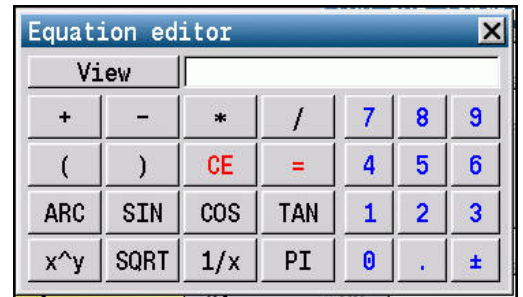
4.5 Program types

The control supports the following programs/contours:

- **Teach-in programs** (cycle programs) are used in **Teach-in** submode
- **smart.Turn** and **DIN main programs** are written in the **smart.Turn** mode of operation
- **DIN subprograms** are written in the **smart.Turn** operating mode and are used in cycle programs and smart.Turn main programs
- **ICP contours** are generated in **Machine** operating mode or in **Teach-in** submode.

  The file name extension depends on the contour described.

In **smart.Turn** operating mode, the contours are saved directly in the main program.

Program type	Folder	Extension
Teach-in programs (cycle programs)	**nc_prog\gtz**	***.gmz**
smart.Turn and DIN main programs	**nc_prog\ncps**	***.nc**
DIN subprograms	**nc_prog\ncps**	***.ncs**
ICP contours	**nc_prog\gti**	
■ Turning contours		■ ***.gmi**
■ Contours of workpiece blanks		■ ***.gmr**
■ Contours on face		■ ***.gms**
■ Lateral surface contours		■ ***.gmm**

4.6 Error messages

Display of errors

The control will display error messages in the following situations:

- Incorrect input
- Logical errors in the program
- Impossible contour elements

When an error occurs, it is displayed in red type in the header. Long and multi-line error messages are displayed in abbreviated form. If an error occurs in a background mode, the error symbol is shown in the operating mode tab. Complete information on all pending errors is shown in the error window.

The control uses the following icons and text colors for different error classes:

Icon	Text color	Error class	Meaning
	Red	Error Prompt	The control displays a dialog with several options you can select from.
	Red	Reset error	The control must be restarted. This message cannot be cleared.
	Red	Error	To continue, you must clear this message. An error message can only be cleared after the cause has been eliminated.
	Yellow	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.
	Blue	Information	You can continue without clearing the message. You can clear the information at any time.
	Green	Note:	You can continue without clearing the message. The control displays the note until you press the next valid key.

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

The error message is displayed in the header until it is cleared or replaced by a higher-priority error.

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

Opening the error window

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

Closing the error window

END	▶ Press the **END** soft key
ERR	▶ Press the **ERR** key. > The control closes the error window.

Detailed error messages

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

Information on the error cause and corrective action:

ERR	▶ Open the error window
↓	▶ Position the cursor on the error message
MORE INFO	▶ Press the **MORE INFO** soft key > The control opens a window with information on the error cause and corrective action.
MORE INFO	▶ Press the **MORE INFO** soft key again to close the window.
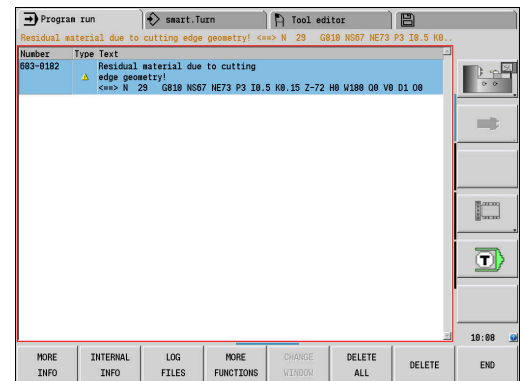
INTERNAL INFO soft key

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

ERR	▶ Open the error window
↓	▶ Position the cursor on the error message
INTERNAL INFO	▶ Press the **INTERNAL INFO** soft key. > The control opens a window with information on the error cause and corrective action.
INTERNAL INFO	▶ Press the **INTERNAL INFO** soft key again to close the window.

GROUPING soft key

If you activate the **GROUPING** soft key, the control displays all warnings and error messages with the same error number in the same line of the error window. This makes the list of messages shorter and easier to read.

To group the error messages:

| ERR | ▶ Open the error window |

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

GROUPING OFF ON	▶ Press the **GROUPING** soft key
	> The control groups identical warnings and error messages.
	> The number of occurrences of the individual messages is indicated in parentheses in the respective line.

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

ACTIVATE SAVING soft key

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

| ERR | ▶ Open the error window |

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

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

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

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

Clearing errors

⚙ The control will automatically clear pending warning or error messages when a new NC program is selected or the previous one is restarted. The machine tool builder specifies in the optional machine parameter **CfgClearError** (no. 130200) whether these messages will be cleared automatically or not.

Clearing errors outside of the error window

| ERR | ▶ Open the error window |

| CE | ▶ Press the **CE** key to clear the errors or information displayed in the header. |

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

Deleting errors

| ERR | ▶ Open the error window |

| ↓ | ▶ Position the cursor on the error message. |

| DELETE | ▶ Press the **DELETE** soft key to clear an individual error. |
| DELETE ALL | ▶ Press the **DELETE ALL** soft key to clear all errors. |

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

Error log

The control stores errors occurred and important events (e.g., system start) in an error log.

Five log files are available. The capacity of these error logs is limited. When a log is full, the control switches to the next log. When the last log is also full, the first is cleared and written to again. If required, switch the log to view the history.

Open the error logs as follows:

| ERR | ▶ Open the error window |

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

| ERROR LOG | ▶ Open the current log file |

| PREVIOUS FILE | ▶ Select previous log file, if needed |

| CURRENT FILE | ▶ Select current log file, if needed |

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

Keystroke log

The control stores each key pressed and important events (e.g., system start) in a keystroke log.

Ten log files are available. The capacity of these keystroke logs is limited. When a log is full, the control switches to the next log. When the last log is also full, the first is cleared and written to again. If required, switch the log to view the history.

Open the keystroke log as follows:

ERR	▶ Open the error window
LOG FILES	▶ Press the **LOG FILES** soft key
ERROR LOG	▶ Open protocol
PREVIOUS FILE	▶ Select previous log file, if needed
CURRENT FILE	▶ Select current log file, if needed

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

Overview of the keys and soft keys for viewing the log

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

Saving service files

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

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

If you repeat the **SAVE FILES** function with the same file name, the previously saved group of service files will be overwritten. Therefore, use a different file name when re-executing the function.

Saving service files

| ERR | ▶ Open the error window |

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

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

| OK | ▶ Press the **OK** soft key |
| | > The control saves the service file. |

4.7 TURNguide context-sensitive help

Application

> **i** Before you can use the TURNguide, you need to download the help files from the HEIDENHAIN home page.
>
> **Further information:** "Downloading current help files", Page 96

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

> **i** The control tries to start the TURNguide in the language that you have selected as the conversational language on your control. If the files with this language are not yet available on your control, the control automatically opens the English version.

The following user documentation is available in the TURNguide:

- User's Manual (**BHBoperating.chm**)
- smart.Turn and DIN programming (**BHBsmartturn.chm**)
- List of All Error Messages (**errors.chm**)
- User's Manual for the **TNCdiag** application, if necessary (**TNCdiag.chm**)

In addition, the **main.chm** "book" file is available, with the contents of all existing .chm files.

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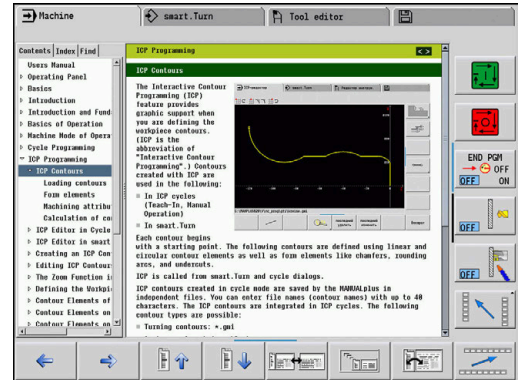
Working with the TURNguide

Calling the TURNguide

There are several ways to start the TURNguide:

[i]

▶ Press the **Info** key if the control is not already showing an error message

▶ First click the help symbol in the lower right-hand corner of the screen, then click the appropriate soft key

[i]   If one or more error messages are waiting for your attention, the control shows the help directly associated with the error messages. To start the TURNguide, you first have to acknowledge all error messages.

When the help system is called on the programming station, the control starts the internally defined standard browser (usually the Internet Explorer), or otherwise a browser adapted by HEIDENHAIN.
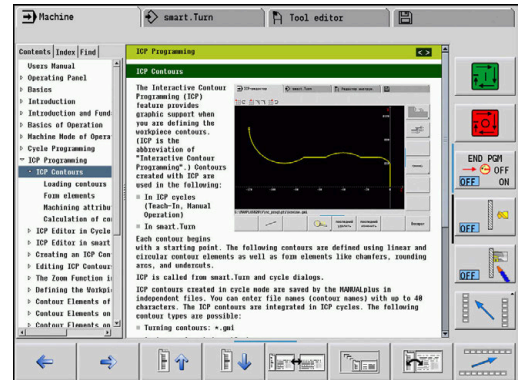
For many soft keys there is a context-sensitive call through which you can go directly to the description of the soft key's function. This functionality requires using a mouse.

Proceed as follows:

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

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

> The cursor turns into a question mark.

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

> The control opens the TURNguide.

> If no specific part of the help is assigned to the selected soft key, the control opens the book file, **main.chm,** in which you can use the full text search function or the navigation to find the desired explanation manually.

Even while you are editing a cycle, context-sensitive help is available:

▶ Select any cycle

[i]

▶ Press the **Info** key

> The control starts the help system and shows a description for the active function (does not apply to miscellaneous functions or cycles that were integrated by your machine tool builder).

Navigating in the TURNguide

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

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

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

> ℹ The key functions described below are available only on the control hardware, and not on the programming station.

Operating element	Function
↑ ↓	■ If the table of contents at the left is active: Select the entry above or below the current one. ■ If the text window at right is active: Move the page downward or upward if texts or graphics are not shown completely
→	■ If the table of contents at the left is active: Expand the current entry or, if it is fully expanded, move to the window at the right. ■ If the text window at right is active: No function
←	■ If the table of contents at left is active: Close a branch of the table of contents ■ If the text window at right is active: No function
ENT	■ If the table of contents at the left is active: Display the selected page. ■ If the text window at right is active: If the cursor is on a link, jump to the linked page
🗐	■ If the table of contents at left is active: Switch the tab between the display of the table of contents, display of the subject index, and the full-text search function and switching to the screen half at right ■ If the text window at right is active: Jump back to the window at left
🗐↓ 🗐↑	■ If the table of contents at the left is active: Select the entry above or below the current one. ■ If the text window at right is active: Jump to next link
←	Select the page last shown

Operating element	Function
	Page forward if you have used the **Select page last shown** function multiple times.
	Move up by one page
	Move down by one page
	Display or hide table of contents
	Switch between full-screen display and reduced display. With the reduced display, you can still see parts of the control window.
	The focus is switched internally to the control application so that you can operate the control when the TURNguide is open. If the full screen is active, the control reduces the window size automatically before changing the focus.
EXIT	Exit TURNguide

Subject index

The most important keywords are listed in the index (Index tab). Select the desired keyword directly by clicking it or by highlighting it with the arrow keys.

The left side is active:

▶ Select the **Index** tab

▶ Activate the **Keyword** input field

▶ Enter the search word

> The control synchronizes the keyword index based on the text you entered and creates a list in which you can find the keyword more easily.

▶ Alternative: Use the **arrow key** to highlight the desired keyword

ENT ▶ Use the **ENT** key to display information on the selected keyword.
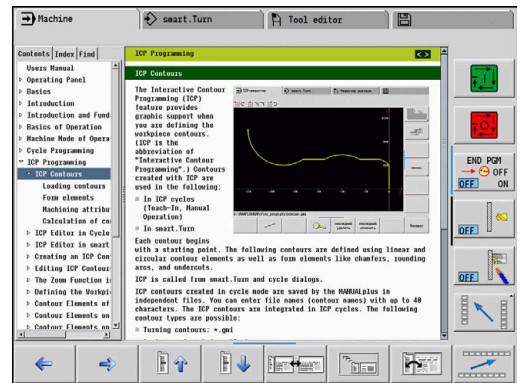
ⓘ You can enter the search word only with a keyboard connected via USB.

Full-text search

In the Find tab you can search the entire TURNguide for a specific word.

The left side is active:

▶ Select the **Find** tab

▶ Activate the **Find:** input field

ENT ▶ Enter the search word

▶ Press the **ENT** key

> The control lists all sources containing the word.

▶ Use the arrow key to highlight the desired source

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

ⓘ You can enter the search word only with a keyboard connected via USB.

The full-text search only works for single words.

If you activate the **Search only in titles** function (by mouse or by pressing the keys), the control searches only through headings and ignores the body text.

Downloading current help files

You'll find the help files for your control software on the HEIDENHAIN homepage **www.heidenhain.de**.

Help files for most conversational languages are at:

▶ Documentation
▶ Manuals
▶ Product, e.g. MANUALplus 620
▶ NC software number (e.g., 548431-xx)

> **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.

▶ Download the compressed CHM file in the desired language and extract it
▶ Move the extracted CHM files to the **TNC:\tncguide\en** directory and the respective language subdirectory on the control

> **ℹ** If you want to use TNCremo to transfer the .chm files to the control, in the connection configuration on the mode form you must select the third option in the transfer area in binary mode.

Language	Directory in TNCremo
German	**TNC:\tncguide\de**
English	**TNC:\tncguide\en**
Czech	**TNC:\tncguide\cs**
French	**TNC:\tncguide\fr**
Italian	**TNC:\tncguide\it**
Spanish	**TNC:\tncguide\es**
Portuguese	**TNC:\tncguide\pt**
Swedish	**TNC:\tncguide\sv**
Danish	**TNC:\tncguide\da**
Finnish	**TNC:\tncguide\fi**
Dutch	**TNC:\tncguide\nl**
Polish	**TNC:\tncguide\pl**
Hungarian	**TNC:\tncguide\hu**
Russian	**TNC:\tncguide\ru**
Chinese (simplified)	**TNC:\tncguide\zh**
Chinese (traditional)	**TNC:\tncguide\zh-tw**

Language	Directory in TNCremo
Slovenian	**TNC:\tncguide\sl**
Norwegian	**TNC:\tncguide\no**
Slovak	**TNC:\tncguide\sk**
Korean	**TNC:\tncguide\kr**
Turkish	**TNC:\tncguide\tr**
Romanian	**TNC:\tncguide\ro**

4.8 DataPilot programming station

Application

Intended for use with the **CNC PILOT 640** and **MANUALplus 620** controls, respectively, the PC programs DataPilot CP 640 and DataPilot MP 620 enable you to create NC programs on a PC, test them before execution, transfer them to the control, and archive them after production is concluded.

DataPilot **can be used for** shop-floor programming as well as for office and off-location program creation and testing. Its practicality and wide range of functions also make DataPilot ideal for training purposes.

Operation

DataPilot is operated through function keys and numerical keys of the PC keyboard.

> **ℹ** For more information about installation and operation, please refer to the DataPilot installation and operating instructions.

4.9 HEROS menu

The **HEROS menu** includes all additional applications available to the control on the **HEROS** operating system. Most of these functions are available to all users of the control. Specific functions, such as the **REMOTE DESKTOP MANAGER**, must be enabled via options.

Further information: "HEROS Functions", Page 683

You have the following options for opening the **HEROS menu**:

- Press the **DIADUR** key
- Go to the **Service** menu and select the **HEROS** icon

5

Operating the Touchscreen

5.1   Display unit and operation

Touchscreen

| ⚙ | Refer to your machine manual.
This function must be enabled and adapted by the machine manufacturer. |

15.6" touchscreen

The touchscreen is distinguished by a black frame and the lack of soft-key selection keys.

1   Header

When the control is on, the screen displays the selected operating modes in the header. To switch to the desired operating mode, tap on that operating mode in the header.

2   Soft-key row for the machine tool builder

3   Soft-key row

The control shows further functions in a soft-key row. The active soft-key row is shown as a blue bar.

4   Screen toggle key for the display of help graphics in cycle programming

5   Call the TURNguide
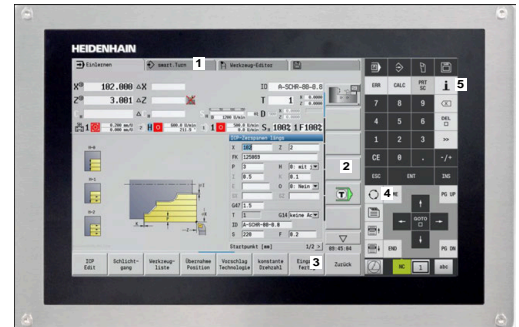
Operation and cleaning

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 entries.

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

Further information: "Touchscreen Cleaning", Page 108

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. |

Operating panel

Basic operation
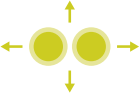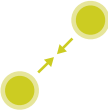
The following keys, for example, can easily be replaced by hand gestures:

Key	Function	Gesture
▷	Shift the soft-key row	Swipe horizontally over the soft-key row
	Soft-key selection keys	Tap on the function in the touchscreen

5.2 Gestures

Overview of possible gestures

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

Symbol	Gesture	Meaning
	Tap	A brief touch by a finger on the screen
	Double tap	Two brief touches on the screen
	Long press	Continuous contact of fingertip on the screen
		ℹ️ If you do not stop holding, the control will automatically cancel the holding gesture after approximately ten seconds. Permanent actuation is thus not possible.
	Swipe	Flowing motion over the screen
	Drag	A combination of long-press and then swipe, moving a finger over the screen when the starting point is clearly defined
	Two-finger drag	A combination of long-press and then swipe, moving two fingers in parallel over the screen when the starting point is clearly defined
	Spread	Two fingers long-press and move away from each other
	Pinch	Two fingers move toward each other

Navigating in the table and NC programs

You can navigate in an NC program or a table as follows:

Symbol	Gesture	Function
	Tap	Mark the NC block or table line
		Stop scrolling
	Double tap	Activate the table line
		Edit NC block or unit
	Swipe	Scroll through the NC program or table
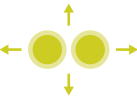
Operating the Simulation

The control offers touch operation in the following cases:

- Programming graphics in the **smart.Turn** operating mode
- Contour reference in the **smart.Turn** operating mode, e.g., **NS** and **NE**
- 3D view in the **Simulation** submode
- 2D view in the **Simulation** submode
- 2D view in the **ICP editor** submode

Rotating, zooming, or moving a graphic

The control supports the following gestures:

Symbol	Gesture	Function
	Tap	Select the contour element
	Double tap	Set the graphic to its original size
	Drag	Rotate the graphic (only 3-D graphics)
	Drag	Adjust the detail (only 2-D graphics, zoom function)
	Two-finger drag	A combination of long-press and then swipe, moving two fingers in parallel over the screen when the starting point is clearly defined
	Spread	Magnify the graphic
	Pinch	Reduce the graphic

Using the HEROS menu

To use the **HEROS menu** menu:

Symbol	Gesture	Function
	Tap	Select the application
	Long press	Open the application

5.3   Functions in the taskbar

Touchscreen Configuration

With the **Touchscreen Configuration** function you can define the properties of the screen.

Adjusting sensitivity

Proceed as follows to adjust the sensitivity:

▶ Press the **DIADUR** key to open the **HEROS menu**

▶ Or go to the **Service** menu to select the **HEROS menu**

▶ Select the **Touchscreen Configuration** menu item

> The control opens a pop-up window.

▶ Select the sensitivity

▶ Confirm with **OK**

Display of the touch points

Proceed as follows to show or hide the touch points:

▶ Press the **DIADUR** key to open the **HEROS menu**

▶ Or go to the **Service** menu to select the **HEROS menu**

▶ Select the **Touchscreen Configuration** menu item

> The control opens a pop-up window.

▶ Select the **Show Touch Points** display

  ◾ **Disable Touchfingers** to hide the touch points

  ◾ **Enable Single Touchfinger** to show the touch point

  ◾ **Enable Full Touchfingers** to show the touch points of all fingers involved

▶ Confirm with **OK**

Touchscreen Cleaning

With the **Touchscreen Cleaning** function you can lock the screen in order to clean it.

Activating the cleaning mode

Proceed as follows to activate the cleaning mode:

▶ Press the **DIADUR** key to open the **HEROS menu**

▶ Or go to the **Service** menu to select the **HEROS menu**

▶ Select the **Touchscreen Cleaning** menu item

> The control locks the screen for 90 seconds.

▶ Clean the screen

If you would like to stop the cleaning mode:

▶ Pull the displayed sliders apart at the same time

6

Machine Mode of Operation

6.1 Machine mode of operation

The **Machine** mode of operation includes all functions for machine setup, workpiece machining, and Teach-in program definition:

- **Machine setup**: For preparations like setting axis values (defining workpiece datum), measuring tools or setting the protection zone
- **Manual mode**: Machine a workpiece manually or semi-automatically
- **Teach-in submode**: "Teach-in" a new cycle program, change an existing program, or graphically simulate cycles
- **Program run submode**: Graphically simulate existing cycle programs or smart.Turn programs and use them for the production of parts

You can produce a part in the usual manner by moving the axes with the handwheels and jog controls, just like on a conventional lathe. In most cases, however, it is much more convenient to use the cycles offered by MANUALplus.

A **Teach-in cycle** is a machining step that has already been programmed for you. This can be any machining operation from a single cut through to a complex machining task like thread cutting. In any case, a cycle is always a complete machining step that is immediately executable once you have defined a few parameters that describe the workpiece to be machined.

In **Machine** mode of operation, cycles are **not saved**. In **Teach-in** submode, each machining step is executed using cycles and then stored and integrated into a complete **teach-in program**. You can subsequently use this **teach-in program** in parts production by repeating it as often as desired in **Program run** submode.

In **ICP programming**, any contour can be defined using linear/circular elements and transition elements (chamfers, rounding arcs, undercuts). You include the contour description in ICP cycles.

Further information: "ICP contours", Page 436

You write **smart.Turn** and **DIN programs** in **smart.Turn** mode of operation. The DIN programming feature provides you with commands for simple traversing movements, DIN cycles for complex machining tasks, switching functions, mathematical operations and programming with variables.

You can either create **independent** programs that already contain all necessary switching and traversing commands and are executed in **Program run** submode, or program **DIN subprograms** that are integrated in teach-in cycles. The commands that you use in a DIN subprogram depend on the task to be performed. DIN subprograms support the complete range of commands that is available for DIN programs.

You can also convert Teach-in programs to smart.Turn programs. This enables you to make use of straightforward Teach-in programming and then optimize or complete the NC program following conversion into **DIN format**.

6.2 Switch-on / Switch-off

Switch-on

> **⚠ 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

The control indicates the status of start-up process. When the control has completed all tests and initializations, it switches to **Machine** mode of operation. The tool display shows the tool that was last used. If errors are encountered during system startup, the control displays the error symbol on the screen. You can check these error messages as soon as the system is ready.

Further information: "Error messages", Page 84

> **ⓘ** After system start, the control assumes that the tool which was last used is still inserted in the tool holder. If this is not the case, you must inform the control of the tool change.

Monitoring of EnDat encoders

If EnDat encoders are used, the control saves the axis positions during switch-off of the machine. During switch-on, the control compares for each axis the position during switch-on with the position saved during switch-off.

If there is a difference, one of the following messages appears:

- **S-RAM error: Saved position of the axis is invalid**: This message is correct if the control has been switched on for the first time, or if the encoder or other control components involved were exchanged
- **Axis was moved after power-off. Position difference: xx mm or degrees**: Check and confirm the current position if the axis was actually moved
- **HW parameters changed: Saved position of the axis is invalid**: This message is correct if configuration parameters were changed

The cause for one of the above listed messages can also be a defect in the encoder or control. Please contact your machine supplier if the problem recurs.

Submode Reference

⚙ Switching on the machine and traversing the reference points can vary depending on the machine tool.

Whether a reference run is necessary or not depends on the encoders used:

- **EnDat encoders**: Reference run is not necessary
- **Distance-coded encoders**: The position of the axes is ascertained after a short reference run
- **Standard encoders**: The axes move to known, machine-based points. As soon as a reference position is traversed, a signal is transmitted to the control. The control knows the distance between the reference position and the machine datum and can now establish the precise position of the axis

Reference run:
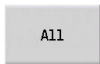
Z	▶ Press the **Z** reference soft key
X	▶ Press the **X** reference soft key
All	▶ Alternative: Press the **All** soft key
⬚	▶ Press the **NC start** key
	❯ The control approaches the reference positions.
	❯ The control activates the position display and activates the **main menu**.
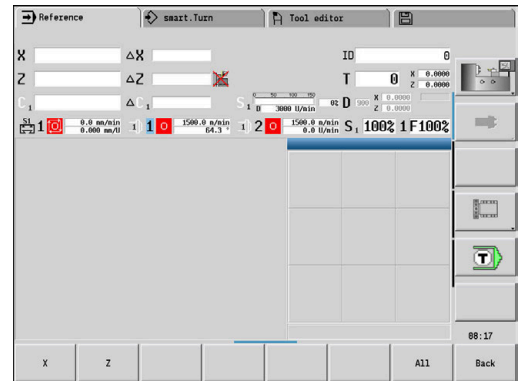
ⓘ In case you traverse the reference points separately for the X and Z axes, you only traverse in either the X or the Z direction.

Switch-off

⚙	Refer to your machine manual. Switch-on and restart can vary depending on the machine tool.

To prevent data from being lost at switch-off, you need to shut down the operating system of the control as follows:

➡	▶ Select the **Machine** operating mode

If an error message is pending:

ERR	▶ Activate the error window

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

OFF	▶ Press the **OFF** soft key

SHUT DOWN	▶ Confirm with the **SHUT DOWN** soft key > The control shuts down.
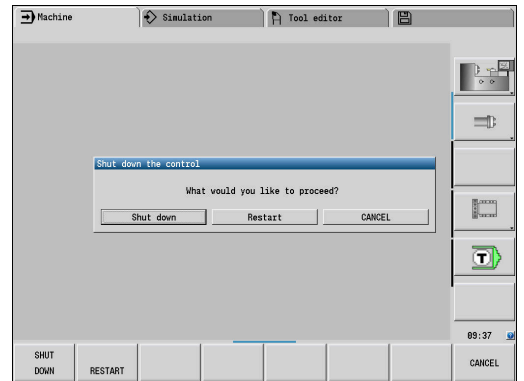
NOTICE

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

---

Restarting the control

Proceed as follows to force a restart:

➡	▶ Select the **Machine** operating mode

OFF	▶ Press the **OFF** soft key

RESTART	▶ Press the **RESTART** soft key > The control restarts.

Integrated Functional Safety (FS)

General information

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).

> ⚙ Refer to your machine manual.
> Your machine tool builder adapts the HEIDENHAIN safety system to your machine.

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).

The HEIDENHAIN safety concepts enables the setup of a system that fulfills the requirements of **Performance Level d, Category 3** in accordance with **DIN EN ISO 13849-1** and **SIL 2 in accordance with IEC 61508 (DIN EN 61508-1)**.

It provides safety-related operating modes, as per **DIN EN ISO 16090-1** (previously DIN EN 12417). Comprehensive operator protection can thus be implemented.

The basis of the HEIDENHAIN safety concept is the dual-channel processor structure, which consists of the main computer (MC) and one or more drive controller modules (CC).

Safety-relevant errors always lead to **safe stopping of all drives** through **defined stop reactions**.

Defined safety functions are triggered and safe operating statuses are achieved via dual-channel safety-relevant inputs and outputs, which have an influence on the process in all operating modes.

In this chapter you will find explanations of the functions that are additionally available on a control with functional safety (FS).

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).

Description	Meaning	Short description
SS0, SS1, SS1D, SS1F, SS2	Safe Stop	Safe stopping of drives using different methods
STO	Safe Torque Off	The power supply to the motor is interrupted. Provides protection against unexpected start of the drives
SOS	Safe Operating Stop	Safe operating stop. Provides protection against unexpected start of the drives
SLS	Safely Limited Speed	Safely limited speed. Prevents the drives from exceeding the specified speed limits when the protective 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

Additional status displays

Machine data display

On a control with functional safety (FS), the status display contains additional information about the current status of the safety functions. The control shows this information as operating states.

The control displays an icon to show the status of the axes:
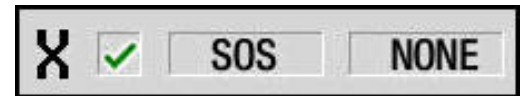
Icon	Short description
✔	The axis has been tested
⚠	The axis has not been tested. All axes must achieve the "tested" status. **Further information:** "Checking the axis positions", Page 117

Safety-related operating mode

The control indicates the active safety-related operating mode by an icon in the vertical soft-key row:

Icon	Safety-related operating mode	Short description
SOM_1	**SOM_1** operating mode active	Safe operating mode 1: Automatic mode, production mode
SOM_2	**SOM_2** operating mode active	Safe operating mode 2: Set-up mode
SOM_3	**SOM_3** operating mode active	Safe operating mode 3: Manual intervention; only for qualified operators
SOM_4	**SOM_4** operating mode active	Safe operating mode 4: Advanced manual intervention, process monitoring, only for qualified users
SOM ⚠	**Warning**: Please refer to your machine manual	

Checking the axis positions

> ⚙ Refer to your machine manual.
> This function must be adapted by your machine manufacturer.

After switch-on, the control checks whether the position of an axis matches the position directly after switch-off. If a deviation occurs, this axis is displayed in red on the position display. Axes that are marked red can no longer be moved while the door is opened.

In such cases you must approach a test position for the axes in question. Proceed as follows:

▶ Select the **Reference** submode

▶ Execute the approach with **NC start** to move the axes in the sequence shown

> The axis moves to the test position.

> Once the test position has been reached, a dialog appears asking whether the test position was approached correctly.

If the control did not approach the test position correctly, proceed as follows:

▶ Confirm with the **END** soft key

If the control approached the test position correctly, proceed as follows:

▶ Confirm with the **OK** soft key

▶ If you confirmed with **OK**, you must confirm the correctness of the test position again with the permissive key on the machine operating panel

▶ Repeat this procedure for all axes that you want to move to the test position

---

NOTICE

Danger of collision!

The control does not automatically check whether collisions can occur between the tool and the workpiece. Incorrect pre-positioning or insufficient spacing between components can lead to a risk of collision while approaching the test positions.

▶ If necessary, move to a safe position before approaching the test positions

▶ Watch out for possible collisions

> ⚙ Refer to your machine manual.
> The location of the test position is specified by your machine tool builder.

Activating feed-rate limits

> ⚙ Refer to your machine manual.
>
> This function must be adapted by your machine manufacturer.

With this function, you can prevent the SS1 reaction (safe stopping of drives) from being triggered when the guard door is opened.

If you press the **F LIMITED** soft key, the control will limit the speed of the axes and of the spindle(s) to the values defined by the machine tool builder. The limitation depends on the safe SOM_x operating mode selected with the aid of the keylock switch.

If SOM_1 is active, the axes and spindles are brought to a stop, because only then will you be allowed to open the guard doors in SOM_1.

▶ Select the **Machine** operating mode
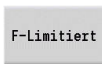
▶ Switch on/off feed rate limit

6.3 Machine data

Input of machine data

In **Machine** operating mode, enter the information for tool, spindle speed, and feed rate/cutting speed in the **TSF** menu (**Set T, S, F** input window).

In the **TSF** menu you also define the maximum speed, the stopping angle and the workpiece material.

Cycle parameters:

- **T**: **Tool number** or **Tool pocket**
- **ID**: **ID no.**
- **F**: **Feed per revolution** or **Minute feed**
- **SP**: **Spindle**
- **S**: **Cutting speed** or **Shaft speed constant**
- **D**: **Maximum speed**
- **A**: **Stopping angle**
- **WS**: **Material**

You can save the cutting data (cutting speed, feed rate) in the technology database as a function of the workpiece material, the tool's cutting material, and the type of machining. With the **Proposed technology** soft key, the data is transferred into the dialog.

If the **Minute feed** soft key is activated, the **F** value is evaluated in **[mm/min]**.

If the **Constant spindle speed** soft key is activated, the **S** value is evaluated in **[rev/min]**.

In Teach-in programs the tool information and technology data are included in the cycle parameters, and in smart.Turn programs they are part of the NC program.

Soft keys for Set T, S, F

Tool correct.	**Further information:** "Tool compensation", Page 162
Measure tool	**Further information:** "Touch-off", Page 159
Tool List / Turret list	Call tool list or turret list **Further information:** "Pocket table setup", Page 129
Proposed technology	Transfer of cutting speed and feed rate from the technology data
Minute feedrate	- **On**: Feed per minute (mm/min) - **Off**: Programmed feed rate per revolution (mm/rev)
Constant speed	- **On**: Constant speed (rpm) - **Off**: Constant surface speed (m/min)
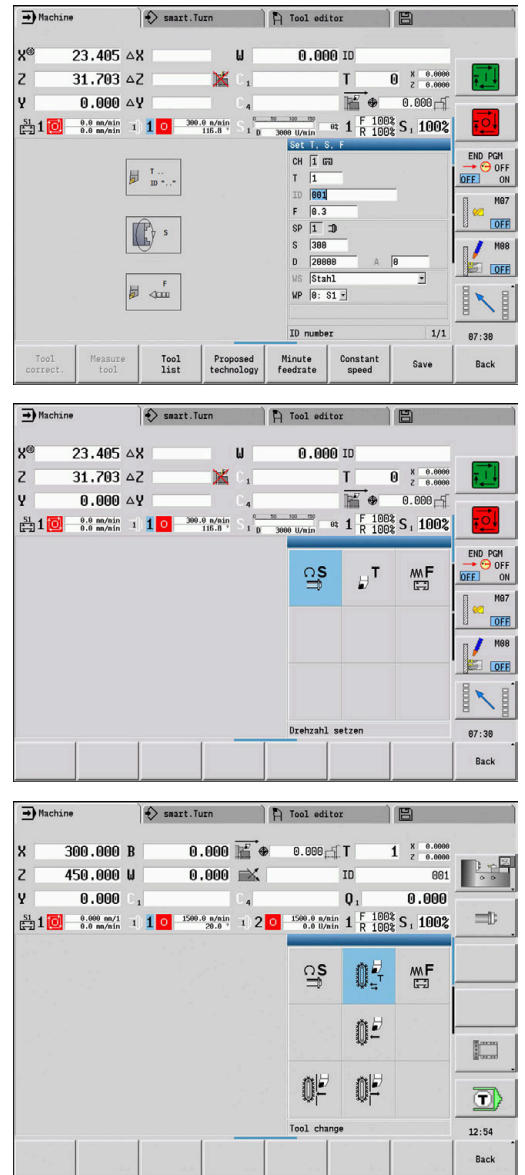
Machine-dependent variants of the TSF dialog

Machine operating panel with spindle change key

If the manufacturer has equipped your machine operating panel with a spindle change key, specify per key for which spindle the entries for **S**, **D** and **A** apply. The **SP** field displays the number of the selected spindle in the **TSF** menu.

$\mathbf{i}$ In the **separateTSFDlg** (604906) machine parameter, you can define how you want the **TSF** dialog to be displayed on machines with turrets:

- **TSF** dialog with input of all cutting data
- Separate dialogs for **T**, **S** and **F**

For machines with tool magazine, separate dialogs are automatically available in the **TSF** menu.

Machine operating panel without spindle change key

For machines with only one main spindle the entries **S**, **D** and **A** always apply to the main spindle.

For machines with one main spindle and one tool spindle, the entries, depending on the inserted tool, apply to the main or tool spindle:

- **Without driven tool**: The parameters **S**, **D** and **A** apply to the main spindle
- **With driven tool**: The parameters **S**, **D** and **A** apply to the selected spindle

Machine with counter spindle and/or B axis

Depending on the machine design, the TSF dialog may contain further information for controlling a counter spindle and/or B axis.

Additional cycle parameter for counter spindle:

- **WP**: **No. of spindle** (machine-dependent)

Additional cycle parameters for B axis:

- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)

TSF entry via a form

> ℹ️ For machines with a tool magazine, separate dialogs are automatically available.

Entering the tool data and technology data:

> ▶ Select **Set T, S, F** (is only selectable in **Machine** mode of operation)
>
> ▶ Define the parameters
>
> ▶ Press the **Save** soft key

NOTICE

Danger of collision!

Depending on the machine, the input of data in the **T** dialog might cause a tilting movement of the tool carrier (e.g. turret). There is a danger of collision during the tilting movement!

> ▶ Move the tool or the tool carrier to a safe position before entering data

---

TSF entry via separate forms

Entering the tool data or technology data:

> ▶ Select **Set T, S, F** (is only selectable in **Machine** mode of operation)
>
> ▶ Select **T** for tool change
>
> ▶ Alternative: Select **S** for spindle speed
>
> ▶ Alternative: Select **F** for feed rate
>
> ▶ Enter the parameters in the submenu
>
> ▶ Press the **Save** soft key

NOTICE

Danger of collision!

Depending on the machine, the input of data in the **T** dialog might cause a tilting movement of the tool carrier (e.g. turret). There is a danger of collision during the tilting movement!

> ▶ Move the tool or the tool carrier to a safe position before entering data

---

Machine with counter spindle

You have to select a workpiece spindle depending on the machine.

If your machine is equipped with a counter spindle, the **WP** parameter is shown in the **TSF** dialog.

Cycle parameters:

- **WP**: **No. of spindle** (machine-dependent)

The **WP** parameter allows you to select the workpiece spindle for machining in **Teach-in** submode and with the MDI cycles of **Machine** mode of operation.

Select the workpiece spindle for machining with **WP**:

- Main drive
- Counter spindle for rear-face machining

The **WP** parameter setting is saved in the Teach-in and MDI cycles and displayed in the corresponding cycle form.

If you selected the counter spindle for rear-face machining with the **WP** parameter, the cycle is mirrored (in the opposite Z direction). Use tools with suitable tool orientation.

> **ⓘ**   In the TSF menu, the setting for the **WP** parameter is changed when you:
> - Run a cycle with another **WP** parameter setting
> - Select a program in **Program run** submode

Machine with B axis

Machines with B axis allow the tool carrier to be tilted for flexible use of tools during turning and milling. By tilting the B axis and rotating the tool you can bring it into positions that enable you to use one and the same tool to machine in the longitudinal and transverse (or radial and axial) directions on the main and counter spindles. In this way, you need fewer tools and fewer tool changes.

Tool data: All tools are described in the tool database by specifying the X, Z and Y dimensions as well as the compensation values. These dimensions and tool orientation must be entered referenced to the **tilt angle B=0°** (reference position).

Cycle parameters:

- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)

The **BW** and **CW** parameter settings are saved in the Teach-in and MDI cycles and displayed in the corresponding cycle form.

NOTICE

Danger of collision!

Depending on the machine, the input of the additional tool parameters might cause a tilting movement of the tool carrier (e.g. turret) or the B axis, and a rotation of the tool. There is a danger of collision during the tilting movement and the rotation!

▶ Move the tool or the tool carrier to a safe position before entering data

---

Machine data display

⚙️ Refer to your machine manual.

The machine data display varies depending on the machine.

The machine tool builder can configure the machine data display according to his specific requirements.

To switch between multiple displays if configured:

⟳ ▶ Press the **triple-arrow key**

Elements of machine data display

X 57.496	**Position display X, Y, Z, W**: Distance between tool tip and workpiece datum
	■ Axis letter:
	  ■ Black: Axis enable granted
	  ■ White = No axis enable

✕⊕	✕▤	✕╫
Handwheel active (panel-mounted handwheel)	Handwheel active (portable serial handwheel)	Clamping active
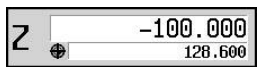
Z −100.000 ⊕ 128.600	**Position display with current datum shift**

C 21.296	**Position display C**: Position of the C axis
	■ Empty field: C axis is not active
	■ Axis letter:
	  ■ Black: Axis enable granted
	  ■ White = No axis enable

X A 11.085	**Display settings of the position display**: Can be set via the **axesDisplayMode** (604803) machine parameter. The setting is shown by a letter next to the position window. ■ **A**: Actual value (setting: **RFACTL**) ■ **N**: Nominal value (setting: **REF NOML**) ■ **L**: Servo lag (setting: **LAG**) ■ **D**: Distance to go (setting: **DIST.**)
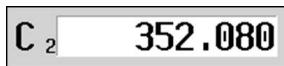
C 2 352.080	**Display of the C axis with the associated spindle number**: The index of the axis letter C indicates the spindle number The numeral is only displayed if an axis was configured multiple times, e.g. a second C-axis is configured as a counter spindle.
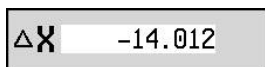
△X −14.012	**Distance-to-go display X, Y, Z, W**: The distance remaining from the current position to the target position of the active traversing command

△Z ✖	**Distance-to-go and protection zone status**: Distance-to-go display and display of status of protection zone monitoring

📟	✖
Protection zone monitoring active	Protection zone monitoring not active

Elements of machine data display
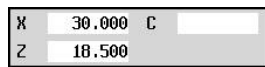
X 30.000 C Z 18.500	**Position display for four axes**: Display of position values for up to four axes The displayed axes depend on the machine configuration.
T 5 X 0.5500 Z 0.6600	**Tool number display**: ■ Tool number of the inserted tool ■ Tool compensation values **For all of the T displays**: ■ **T** highlighted in color: Driven tool ■ Tool number or **ID** highlighted in color: Mirrored tool holder ■ Tool number with index: Multipoint tool ■ The letter **X**/**Z** of the compensation highlighted in color: Special compensation is active in the X and Z direction
T 045 X 0.000 Z 0.000	**T ID display**: ■ **ID** of the inserted tool ■ Tool compensation values
T Stechwerkzeug222	**T ID display without compensation values**: ■ **ID** of the inserted tool
D X 0.2200 Y 0.0000 Z 5.1000 S 5.1000	**Tool compensation**: ■ Special compensation only for recessing tools or button tools ■ Special compensation value in gray: Special compensation is not active ■ The letter **X**/**Z** of the compensation highlighted in color: Special compensation is active in the X and Z direction
D 901 X 0.5000 Z 0.3000	**Additive compensation**: ■ Compensation values in gray: D compensation is not active ■ Compensation values in black: D compensation is active
T MT RT MZ RZ	**Tool life information**: ■ **T**: 　■ Black = global tool life monitoring is on 　■ White = global tool life monitoring is off ■ **MT**, **RT** active: Monitoring according to tool life ■ **MZ**, **RZ** active: Monitoring according to part quantity ■ All fields empty: Tool without tool-life monitoring
1 100% 10394.1 mm/min	**Slide display and cycle status**: ■ Upper field: Setting of the override control ■ Lower field with white background: Actual feed rate ■ Lower field with gray background: Programmed feed rate with stationary slide
1 6.789 mm/1 6.779 mm/1	**Slide display and cycle status**: ■ Upper field: Programmed feed rate ■ Lower field: Actual feed rate
1 100% 6.789 mm/1 6.779 mm/1	**Slide display and cycle status**: ■ Upper field: Setting of the override control ■ Middle field: Programmed feed rate ■ Lower field: Actual feed rate
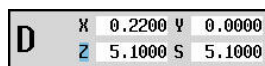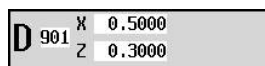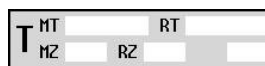
Elements of machine data display

`S4` `1` `[O]` `100%` `2.540 mm/1` `0.000 mm/1`	**Slide display with rear-face machining** : ■ If rear-face machining is enabled, the slide icon is highlighted in blue

`1` `M3` `100%` `2091.6 1/min`	**Spindle display with spindle number, gear range and spindle status**:
	■ Upper field: Setting of the override control
	■ Lower field: Actual speed or spindle position
	For all of the spindle displays:
	■ Spindle symbol:
	  ■ Black: Spindle enable granted
	  ■ White = No spindle enable
	■ Numeral in spindle symbol: Gear range
	■ Numeral at right next to the spindle symbol: Spindle number
	■ If a spindle key exists, the number of the selected spindle is highlighted in color
	■ Spindle status: see "Spindle", Page 128
	■ Display of the programmed speed in **rpm** or **m/min**
	■ Display of the actual speed in **rpm**
	■ If **M19** is active and the machine tool builder has made the setting, when the spindle is not turning, the display shows the spindle position instead of the actual spindle speed
	■ If a spindle is in slave mode during synchronous operation, the value **0** is displayed instead of the programmed speed
	■ In synchronous mode, the spindle symbols for both the master and the slave spindle are highlighted in color
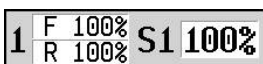
`1` `M3` `150.0 m/min` `107.0 1/min`	**Spindle display with spindle number, gear range and spindle status**: ■ Upper field: Programmed speed ■ Lower field: Actual speed or spindle position If the programmed spindle speed is displayed in red, a limitation is active and the programmed nominal value will not be reached.
`1` `M3` `100%` `150.0 m/min` `107.0 1/min`	**Spindle display with spindle number, gear range and spindle status**: ■ Upper field: Setting of the override control ■ Middle field: Programmed speed ■ Lower field: Actual speed or spindle position If the programmed spindle speed is displayed in red, a limitation is active and the programmed nominal value will not be reached.
`1` `F 100%` `R 100%` `S1 100%`	**Override display of the active spindle**: ■ **F**: Feed rate ■ **R**: Rapid traverse ■ **S**: Spindle
`Z` `0   50   100  150` `0%`	**Motor utilization**: Utilization of the servo drive with respect to the rated torque ■ Digital axis and spindle motors ■ Analog axis and spindle motors, if set up by the machine tool builder
`MP   50 P        2`	**Display of quantity**: The quantity is incremented after each **M30**, **M99** or **M18** programmed counter pulse ■ **MP**: Defined quantity ■ **P**: Number of finished parts
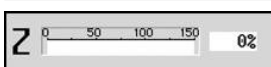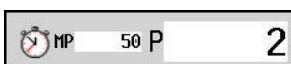
Elements of machine data display
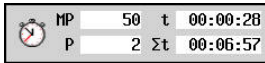
	Display of quantity and job time: The quantity is incremented after each **M30**, **M99** or **M18** programmed counter pulse ■ **MP**: Defined quantity ■ **P**: Number of finished parts ■ **t**: Run time of the current program ■ **Sum t**: Total time
	Display of skip levels and M01 conditional stop: ■ Defined skip levels (upper row) and set or activated skip levels (lower row) ■ Setting for **M01**: M01 is not executed in **Continuous run** mode (yellow display)
	Load monitoring display: ■ Left field: zone number (here: 345) ■ Right fields: monitored axes (max. 4)
	Functional safety (FS) display: ■ Upper graphics: FS status of the axes and spindles ■ Lower graphic: FS status of the axis group (slide icon) and the spindle group (spindle icon) **Further information:** "Integrated Functional Safety (FS)", Page 114
	Display of rear-face machining: Information about rear-face machining is shown in the RSM display (RSM: Rear Side Machining) ■ RSM status When **HDT** mode is active, the control uses the **HDT** icons. ■ Active datum shift of the configured RSM axis
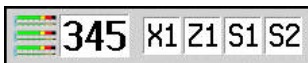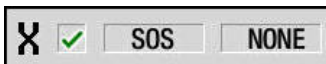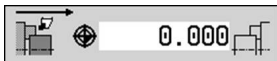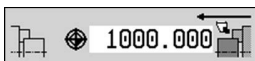
HDT mode is active in front of the workpiece	**HDT** mode is active behind the workpiece

	B-axis display: The information displayed about the status of the plane tilted with **G16** varies depending on the setting of the machine parameters ■ Programmed angular value of the B axis ■ Display of the current **I**, **K**, **U**, and **W** values 　■ **I**: Plane reference in the **X** axis 　■ **K**: Plane reference in the **Z** axis 　■ **U**: Shift in the **X** axis 　■ **W**: Shift in the **Z** axis

Elements of machine data display

A 0.000 B 0.000 C 0.000
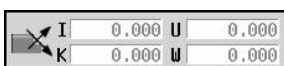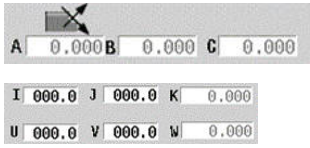 	
I 000.0 J 000.0 K 0.000
U 000.0 V 000.0 W 0.000
	Working plane display: The information displayed about the status of the plane tilted with **G160** varies depending on the setting of the machine parameters
	
	■ Programmed spatial angles for the **A**, **B**, and **C** axes
	
	■ Display of the current **I**, **J**, **K**, **U**, **V**, and **W** values
	
	    ■ **I**: Tilting position in the **X** axis
	
	    ■ **J**: Tilting position in the **Y** axis
	
	    ■ **K**: Tilting position in the **Z** axis
	
	    ■ **U**: Shift in the **X** axis
	
	    ■ **V**: Shift in the **Y** axis
	
	    ■ **W**: Shift in the **Z** axis

26.03.2013 10:24:20	**Date and time display**
MANUALplus 620 HEIDENHAIN	**Display of an integrated logo**

Cycle states

The control shows the current cycle status with the cycle symbol.

Cycle symbols

[▷I]	**Cycle ON** status Cycle or program execution is active	
[○]	**Cycle OFF** status Cycle or program execution is not active	

Axis feed rate

F is the identification letter for feed data.

Depending on the setting of the **Minute feed** soft key, data is entered in:

■ Millimeters per spindle revolution (feed per revolution)

■ Millimeters per minute (feed per minute)

On the screen, you can tell the type of feed rate from the unit of measure in the input field.

You can change the feed value with the **feed compensation controller** (feed override) (range: 0 % to 150 %).

Spindle

S is the identification letter for spindle data.

Depending on the setting of the **Constant speed** soft key, data is entered in:

- Revolutions per minute (constant speed)
- Meters per minute (constant surface speed)

The input range is limited by the maximum spindle speed. You define the speed limitation in the **TSF** dialog window or in DIN programming with the **G26** code. The speed limit remains in effect until a new speed limit value is programmed.

You can change the spindle speed with the speed compensation controller (speed override) (range: 50 % to 150 %).

> 🛈 ■ If you are machining at a constant cutting speed, the control calculates the spindle speed from the position of the tool tip. The smaller the diameter of the tip, the higher the spindle speed. The maximum spindle speed, however, is never exceeded
> ■ The spindle symbols indicate the direction of spindle rotation as seen from the point of view of the machinist
> ■ The spindle designation is set by the machine tool builder

Spindle symbols (S display)

M3	Direction of spindle rotation **M3**
M4	Direction of spindle rotation **M4**
◯	Spindle stop **M5**
M19	Spindle is position-looped **M19**
C	C axis on spindle motor is active

Spindle designations

H	0	1	Main spindle
1	1	2	Driven tool

6.4   Pocket table setup

Tool data such as length and radius, as well as further tool-specific information required by the control for executing diverse functions, are saved in the **toolturn.htt** tool table (in the **TNC:\table** directory). In the control this tool table is designated a **tool list**.

Tools equipped in your tool carriers are saved in the **ToolAllo.tch** pocket table (in the **TNC:\table** directory). This pocket table is made available depending on the machine and designated a turret list or magazine list.

Machine with tool holder (Multifix)

You do not need to manage a pocket table for machines with multifix tool holders because the tool carrier only has one pocket:

- **T**: **Tool number** – turret pocket number (always **T1**)
- **ID**: **ID no.** – Tool name (max. 16 characters)
  Select the tool ID number from the tool list.
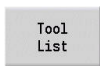
Tool List	▶ Press the **Tool list** soft key
	> The control opens the list.

> ⚙ Refer to your machine manual.
>
> The turret, magazine and multifix tool systems can be used together on one machine.
>
> The machine tool builder defines the number of the multifix pocket.

Machine with turret

Press the **Turret list** soft key to open the list of the current turret assignment. There is a place in the table for every tool holder of the turret (and possibly Multifix). During setup, each tool (ID number) is assigned to a tool holder. In the turret list, multipoint tools are displayed with all cutting edges.

The turret list can be set up through the **TSF** menu or directly from the cycle dialogs in **Teach-in** submode:

- **T**: **Tool number** – turret pocket number
- **ID**: **ID no.** – Tool name (max. 16 characters)
  The tool name is entered automatically.

Cursor in the **TSF** menu in the **T** input field:

Turret list	▶ Press the **Turret list** soft key
	> After opening, the turret list can be edited.

Cursor in the **TSF** menu in the **ID** input field:
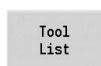
Tool List	▶ Press the **Tool list** soft key
	> The tool list is also opened in addition to the turret list.
	> The turret can be populated with tools from the list.

In the Teach-in cycle, you program the turret pocket as a **T number**. The tool ID number is automatically entered in **ID** if a tool is assigned to the pocket.

⚙ Refer to your machine manual.

The turret, magazine and multifix tool systems can be used together on one machine.

The machine tool builder defines the number of the multifix pocket.

Machine with magazine

Press the **Magazine list** soft key to open the list of the current tool carrier assignments. There is a place in the table for every tool holder. During setup, each tool (ID number) is assigned to a tool holder.

On machines with a magazine, the tool is also changed via the **TSF** menu:

- **T**: **Tool number** – turret pocket number (always **T1**)
- **ID**: **ID no.** – Tool name (max. 16 characters)
  The tool name is entered automatically.

Magazine list	▶ Press the **Magazine list** soft key

The tools used are listed in the magazine list. The magazine can be loaded and unloaded via the **TSF** menu.

> ⚙ Refer to your machine manual.
>
> The turret, magazine and multifix tool systems can be used together on one machine.
>
> The machine tool builder defines the number of the multifix pocket.

Filling the turret list with contents from the tool list

The turret list shows the current assignments of the tool carrier. The turret list can be set up through the **TSF** menu or directly from the cycle dialogs in **Teach-in** submode.

Display the entries in the tool list in order to transfer entries from the list to the turret assignment list. The control displays the tool list in the lower area of the screen. The cursor keys are active in this list. You can move the cursor directly to a tool ID number by entering the first few letters or digits of the ID number.

> ℹ If any content of a table is incompletely displayed, you can change the width of the table columns.
>
> The changes are effective in all operating modes and are persistent across reboots of the control.

Open the turret list as follows:

S ▯T ▯MMF	▶ Select **Set T, S, F** (is only selectable in **Machine** operating mode)
	▶ Alternative: Open the cycle dialog
Tool List	▶ Press the **Tool list** soft key to activate the turret assignment list and the tool list
	▶ Adapt the turret assignment list

Transfer tools from the database as follows:

Pocket down	▶ Select the position in the turret assignment list
	▶ Select and sort the entries in the tool database
↓	▶ Use the cursor keys to select the entry in the tool database
Load tool	▶ Transfer the selected tool to the turret assignment list

Selecting and sorting the entries in the tool database

Tool type	The control opens the soft-key menu for selecting the desired tool type
More filters	The control opens the soft-key menu providing additional filter options
View	The control opens the soft-key menu providing various sorting options
Sort by ID / Type	This soft key is available once you press the **View** soft key. Sorts the tools in the displayed list as desired according to: ■ Tool type ■ Tool ID number ■ Tool orientation Each time you press the soft key, it toggles between the available sorting options.
Reverse sorting	This soft key is available once you have pressed the **View** soft key. Toggles between ascending and descending sorting
Edit the tool	This soft key is available once you press the **Turret list** soft key. Not active here
Back	Closes the tool list

Editing the turret list

The turret assignment list indicates the current assignment of the tool carrier. When you set up a turret list, you enter the ID numbers of the tools.

The turret list can be set up through the **TSF** menu or directly from the cycle dialogs in **Teach-in** submode. You can use the arrow keys to select the desired turret pocket.

The **Turret assignment** list shows the most important tool parameters.
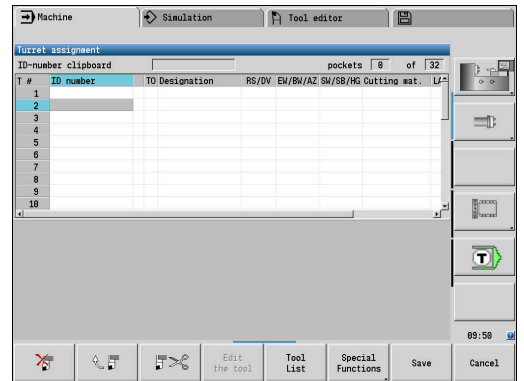
> ℹ️ If any content of a table is incompletely displayed, you can change the width of the table columns.
>
> The changes are effective in all operating modes and are persistent across reboots of the control.

Further information: "General tool parameters", Page 588

You can also set up manual changing systems in the turret assignment list.

Further information: "Setting up the holder for manual change systems", Page 586

Setting up the turret list:

▶ Select **Set T, S, F** (is only selectable in **Machine** mode of operation)

▶ Alternative: Initiate the cycle dialog

▶ Press the **Turret list** soft key to activate the turret assignment list and the tool list

▶ Use the arrow keys to select a turret pocket

▶ Adapt the turret assignment list with the soft keys

▶ Alternative: Enter the tool ID number directly

Entering the tool ID number directly:

▶ Press the **ENT** key to activate direct input

▶ Enter the tool ID number

▶ Press the **INS** key to conclude input

▶ Alternative: Press the **ESC** key to cancel input

Soft keys in turret list

	Delete entry
	Paste entry from clipboard
	Cut out entry and save it in the clipboard
Tool List	Show entries in the tool database
Special Functions	Switch to next menu
Delete all	This soft key is available once you press the **Special functions** soft key. Delete the complete turret list
Set teeth to new	This soft key is available once you press the **Special functions** soft key. Reset tool life
Switch columns	This soft key is available once you press the **Special functions** soft key. Switches the tool parameter view
Set-up the holder	This soft key is available once you press the **Special functions** soft key. Opens the **Table of the tool holder**
Remove tool holder	This soft key is available once you press the **Special functions** soft key. Deletes the tool holder from the turret assignment list
Back	Back by one menu level
Save	Load the tool number and the tool ID number into the TSF or cycle dialog
Cancel	Close the turret list without loading the tool number and tool ID number into the dialog. Changes to the turret list are retained

Editing the magazine list

For machines with tool magazines, the magazine list indicates the current assignment of the existing magazines and tool carriers in the working space. The magazine list can be edited via the **TSF menu**.

> **i** If any content of a table is incompletely displayed, you can change the width of the table columns.
>
> The changes are effective in all operating modes and are persistent across reboots of the control.
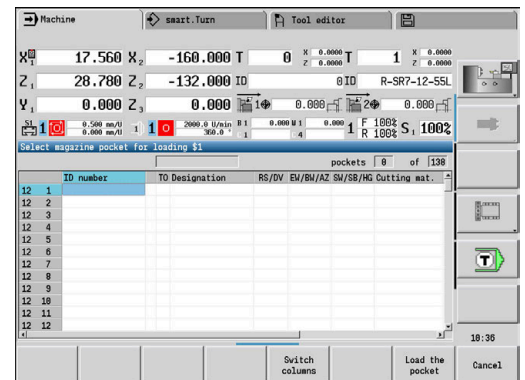
Loading the magazine:

▶ Select **Set T, S, F** (is only selectable in **Machine** mode of operation)

▶ Select **Load magazine**

▶ Press the **Load the pocket** soft key to activate the tool list

▶ Select the tool

▶ Press the **Load tool** soft key to select the tool

▶ Press the **Save** soft key to transfer the tool to the magazine list

Changing the tool:

▶ Select **Set T, S, F** (is only selectable in **Machine** mode of operation)

▶ Select **Tool change**

▶ Press the **Magazine list** soft key to select the tool

▶ Alternative: Enter the tool ID number directly

▶ Press the **Save** soft key to change the tool

Returning the tool to the magazine:

▶ Select **Set T, S, F** (is only selectable in **Machine** mode of operation)

▶ Select **Return tool to the magazine**

▶ Press the **Save** soft key to return the tool to the magazine

Unloading the magazine:

▶ Select **Set T, S, F** (is only selectable in **Machine** mode of operation)

▶ Select **Unload magazine**

▶ Select the tool

▶ Press the **Unload** soft key

▶ Press the **Save** soft key to remove the tool from the magazine list

Tool call

> ⚙ Refer to your machine manual.
> This function is also available on machines with a tool magazine.
> The control will then use the magazine list instead of the turret list.

Parameters for calling the tool

T is the identification letter for the tool holder.

> ⚙ Refer to your machine manual.
> The designation of the tool pockets may vary depending on the machine.
> Each tool holder of a tool carrier has a unique T number in the working space.

ID designates the tool ID number.

> ℹ The ID number of a tool is defined when you create a tool in **Tool editor** mode of operation. Each tool has a unique **ID**.

Variants of the tool call

- A tool holder, e.g. Multifix

  The tool is called by its **ID**. The tool pocket number **T** is always **1**. The control does not maintain a turret list.

- Multiple tool holders, e.g. turret

  The tool is called with **T** (turret pocket number). The ID number **ID** is shown and automatically entered in the dialogs. The control maintains a turret list.

In the turret list, multipoint tools are displayed with all cutting edges.

The parameters for calling the tool can be entered in the **TSF** dialog in **Machine** mode of operation. In **Teach-in** submode and **smart.Turn** mode of operation, **T** and **ID** are cycle parameters.

> ℹ If a **T** number is entered in the **TSF** dialog with an **ID** number that is not defined in the turret list, then the turret list will be changed accordingly. The existing turret list will be overwritten.

Driven tools

◾ Driven tools are defined in the tool description

◾ The driven tool can be operated with feed per revolution if the tool spindle drive is equipped with a rotary encoder

◾ If driven tools are used at a constant cutting speed, the spindle speed is calculated from the tool diameter

Tools in different quadrants

Example

The **principal tool carrier** of your lathe is in front of the workpiece (standard quadrant). An additional tool holder is behind the workpiece.

When the control is configured, it is defined for each tool holder whether the X dimensions and the direction of rotation of circular arcs are mirrored. In the above-mentioned example the additional tool holder is assigned the attribute **mirrored**.

If this method is used, all machining operations are programmed **as usual**—regardless of which tool holder is used for machining. The **Simulation** submode also shows all machining operations in the **standard quadrant**.

The tools are also described and dimensioned for the **standard quadrant**—even if they are inserted in the additional tool holder.

Mirroring does not become effective until the machining of the workpiece, i.e. when the additional tool holder is executing the machining operation.

Tool life monitoring

If desired, you can have the control monitor tool life or the number of parts that are produced with a specific tool.

The tool life monitoring function adds the time a tool is used at **feed rate**. The quantity monitoring counts the number of finished parts. The count is compared with the entry in the tool data.

As soon as the tool life expires or the programmed quantity is reached, the control sets the diagnostic bit 1. This causes an error message to be issued the next time the tool is called and program run will be stopped.

- For teach-in programs, simple tool life monitoring is available

  Here, the control informs you when a tool is worn out.

- With smart.Turn and DIN PLUS programs, you can choose between simple tool life monitoring feature or tool life monitoring with replacement tools (option 10).

  If you use replacement tools, the control automatically inserts the **sister tool** as soon as the tool is worn out. The control does not stop the program run until the last tool of the tool sequence of exchange is worn out.

You can activate / deactivate the tool life management in the **lifeTime** (601801) user parameter.

The control manages the type of monitoring, the tool life/remaining tool life, and the maximum number of pieces/remaining number of pieces in the diagnostic bits of the tool data. You can manage and display the diagnostic bits and the tool life in **Tool editor** mode of operation.

Further information: "Editing tool life data", Page 580

> **i** If a tool is renewed (e.g., replacement of indexable inserts), the tool life and the number of pieces must be reset in **Tool editor** mode of operation.

You can define replacement tools when setting up the tool carrier in **smart.Turn** mode of operation. The **sequence of exchange** can contain more than one sister tool. The sequence of exchange is a part of the NC program.

Further information: smart.Turn and DIN Programming User's Manual
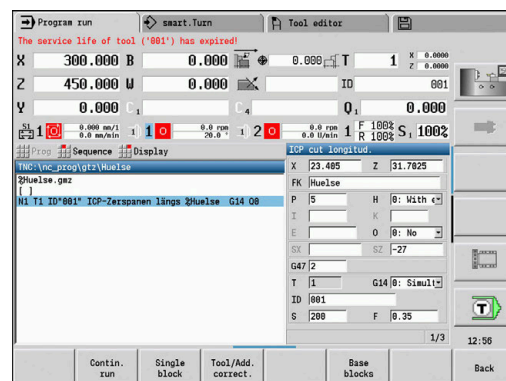
Resetting the tool life in the turret list

Resetting the tool life:

▶ Select **Set T, S, F** (is only selectable in **Machine** mode of operation)

▶ Press the **Turret list** soft key

▶ Press the **Special functions** soft key

▶ Press the **Set teeth to new** soft key

▶ Press the **YES** soft key to confirm the confirmation prompt

▶ Press the **Back** soft key

Resetting the tool life in the magazine list

Resetting the tool life:

▶ Select **Set T, S, F** (is only selectable in **Machine** mode of operation)

▶ Select **Tool change**

▶ Press the **Magazine list** soft key
▶ Select the tool
▶ Press the **Edit the tool** soft key

▶ Press the **New tooth** soft key

▶ Press the **Back** soft key

6.5   Machine setup

The machine always requires a few preparations, regardless of whether you are machining a workpiece manually or automatically.

In **Machine** operating mode, you can call the following functions through the **Setup** menu item:

- **Set axis value** (define workpiece datum)
  - **Machine reference** (homing the axes)
- **Set protection zone**
- **Set tool change point**
- **Set C-axis values**
- **Set machine dimensions**
- **Dyna. Coll. Monitoring (DCM)**
- **Service**
  - **Display operating times**
  - **Set up touch probes**
  - **Adjust the system time**
  - Start **TNCdiag**
  - **HEROS menu**
- **Probing**
  - **Calibrate the touch probe**

Defining the workpiece datum

In the dialog, the distance between the machine datum and the workpiece datum (also known as **offset**) is shown as **XN** and **ZN**. If the workpiece datum is changed, the display values will be changed accordingly.

> 🛈 The workpiece datum can also be set in the Z axis using a touch probe. When setting the datum, the control checks which type of tool is currently active. If you select the Workpiece datum setup function and a touch probe is inserted, the control automatically adjusts the input form. Press **NC start** to start the measuring process.
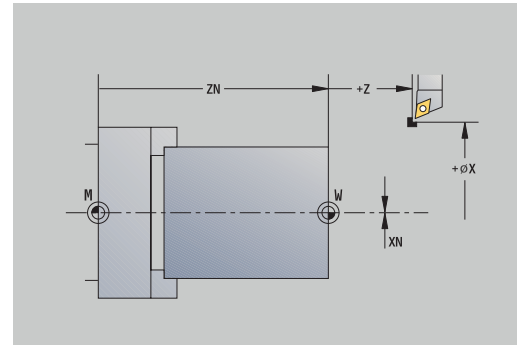
Setting the workpiece datum:

▶ Select **Setup**

▶ Select **Set axis value**
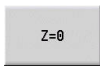
▶ Touch the workpiece datum (end face)

Z=0 ▶ Define this point as the workpiece datum **Z = 0**

▶ Alternative: Enter the distance between the tool and the workpiece datum as **Measuring pt. coordinate Z**

> The control will calculate the workpiece datum **Z**

Delete Z offset ▶ Alternative: Machine datum Z = workpiece datum Z (offset = 0)

Absolute offset ▶ Alternative: This makes it possible to enter the datum shift directly in **ZN**.

Save ▶ Press the **Save** soft key

Defining offsets

Before using datum shifts with **G53**, **G54** and **G55**, you need to define the offset values in setup mode.

Setting an offset:

▶ Select **Setup**

▶ Select **Set axis value**
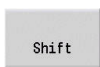
Shift ▶ Press the **Shift** soft key

▶ Press the **G53**, **G54** or **G55** soft key

Save ▶ Press the **Save** soft key

> The control saves the values to a table. In this way, you can activate the offsets in the program by entering the respective **G** codes
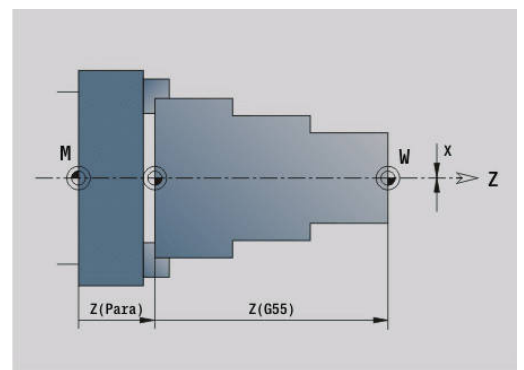
Axis reference run

It is possible to home axes that have already been homed. You can select individual axes or all axes simultaneously.

Homing:

| Machine reference | ▶ | Press the **Machine reference** soft key |

| X | ▶ | Press the **Z** reference and **X** reference soft key |

| All | ▶ | Alternative: Press the **All** soft key |

	▶	Press the **NC start** key
	>	The control automatically traverses the reference marks.
	>	The control refreshes the position display.
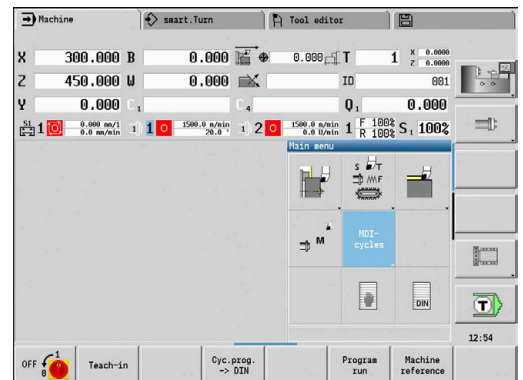
Setting the protection zone

With active protection zone monitoring, the control checks for every movement whether the **protection zone in −Z direction** would be violated. If it detects such a violation, it stops the axis movement and generates an error message.

The **Set protection zone** setup dialog shows the distance between the machine datum and the protection zone in **-ZS**.

The status of the protection zone monitoring is shown in the machine display if it has been configured by the machine manufacturer.

Setting the protection zone and switching off the monitoring function:
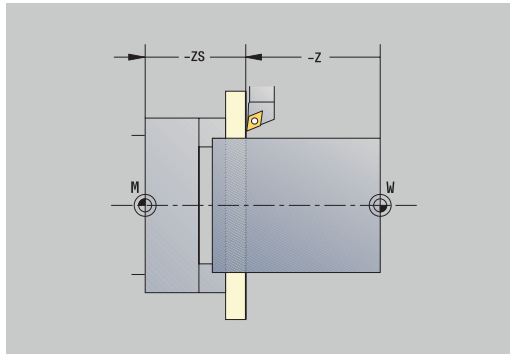
▶ Select **Setup**

▶ Select **Set protection zone**
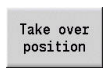
▶ Move the tool with the axis keys or the handwheel until it reaches the **protection zone**

Take over position

▶ Use the **Take over position** soft key to load this position as protection zone

▶ Alternative: Enter the position of the protection zone relative to the workpiece datum (**Measuring pt. coordinate -Z** field)

Save

▶ Use the **Save** soft key to load the entered position as protection zone

Protectn off

▶ Alternative: Switch off protection zone monitoring

> **ⓘ** ■ If the **Set protection zone** input window is open, protection zone monitoring is inactive
> ■ In DIN programming, you deactivate protection zone monitoring with **G60 Q1** and reactivate it with **G60**

Protection zone status

	Protection zone monitoring active
	Protection zone monitoring not active

Defining the tool change position

With the cycle **Move to tool change position** or the DIN command **G14**, the slide moves to the **tool change position**. Define the tool change position such that the turret can rotate without collision and the tools do not damage the workpiece during tool change.

Defining the tool change position:

▶ Select **Setup**

▶ Select **Tool change point**

▶ Move to the tool change position using the axis keys or the handwheel.

Take over position ▶ Press the **Take over position** soft key
> The control saves the current position as the tool change position.
▶ Alternative: Enter the tool change position directly
▶ Enter the desired tool change position in the **X** and **Z** input fields in machine coordinates (**X** = radius dimension)
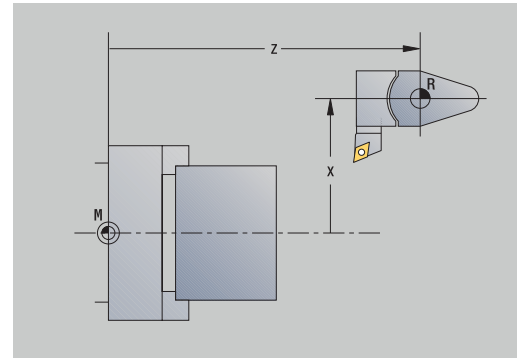
ℹ The coordinates of the tool change position are entered and displayed as the distance between the machine datum and the point of reference on the tool carrier. It is recommended that you move to the tool change position and load the position with the **Take over position** soft key.

Setting C-axis values

The **Set C-axis values** function enables you to define a datum shift for the workpiece spindle:

- **CN**: **Datum shift in C axis** – Position value of the workpiece spindle
- **C**: **Datum shift in C axis**
- **CM**: **Measuring pt. coordinate** – Set current position to the defined value

Setting the datum of the C axis:

▶ Select **Setup**

▶ Select **Set C-axis values**

▶ Position the C axis

▶ Define the position as the C-axis datum
▶ Alternative: Set the current position to the defined value
▶ Press the **Absolute offset** soft key

▶ Enter the value in the **CM** input field
▶ Enter the datum shift of the C axis
▶ Confirm your input
> The control calculates the datum of the C axis.
▶ Alternative: Delete the datum shift of the C axis

Expanded form view for machines with counter spindle

If your machine is equipped with a counter spindle, the **CA** parameter is shown. The **CA** parameter enables you to specify for which workpiece spindle (main spindle or counter spindle) entries for the **Set C-axis values** function will be effective.

The active angle offset is shown in the **CV** parameter. An angle offset is activated with **G905** to match the position of main and counter spindle to each other. This may be necessary if both spindles need to be synchronized for a part transfer. The **Delete CV offset** soft key enables you to reset an active angle offset.

Additional parameters for machines with counter spindle:
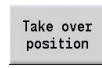
- **CV**: **Datum shift in C axis** – Active angular offset
- **CA**: **No. of C axis** – Selection of C axis (main spindle or counter spindle)
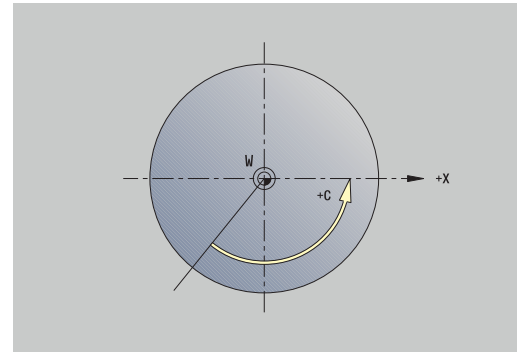
Setting up machine dimensions

The **Set machine dimensions** function allows you to save any positions for use in NC programs.

Setting up machine dimensions:

▶ Select **Setup**

▶ Select **Set machine dimensions**

▶ Enter the number for the machine dimensions

▶ Transfer the position of a single axis as machine dimension

▶ Alternative: Transfer the position of all axes as machine dimensions

▶ Save the machine dimensions
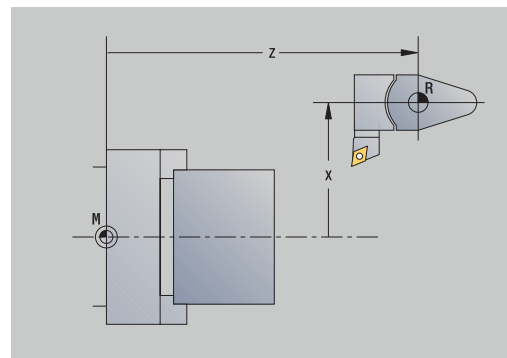
Incremental jog positioning

> Refer to your machine manual.
> Your machine manufacturer enables this function.

With incremental jog positioning, the control moves a machine axis by a defined distance with every activation of the axis key. This defined distance is called increment.

The input range for the infeed is from 0.001 mm to 10 mm.

Parameters:

- **SMA**: **Active jog increment** – currently selected jog increment
- **SM**: **Jog increment** – input field for jog increment

Applying a user-defined jog increment

For incremental jog positioning, proceed as follows:

▶ Select the **Machine** operating mode

▶ Select the **Setup** menu item

▶ Select the **Jog increment** menu item

▶ Enter the required jog increment in the **SM** field

▶ Press the **Save** soft key

> The control displays the defined jog increment in the **SMA** field.

▶ Press the **Jog inc. on** soft key

> The control activates incremental jog positioning

▶ Press the axis key

> The control positions incrementally.

> Press the **Jog inc. on** soft key again to deactivate incremental jog positioning.

Applying a pre-defined jog increment

The **Jog increment** menu item provides soft keys for pre-defined jog increments. When you press one of these soft keys, the corresponding jog increment automatically appears in the **SMA** field.

Soft key	Function
0.001	Jog increment 0.001 mm
0.01	Jog increment 0.01 mm
0.1	Jog increment 0.1 mm

Setting up touch probes

For configuring and managing touch probes, the **Set up touch probes** menu is available.

To open the **Set up touch probes** menu:

▶ Select **Setup**

▶ Select **Service**

▶ Select **Set up touch probes**
▶ The control opens the **Device configuration** window in the **Set up touch probes** menu.
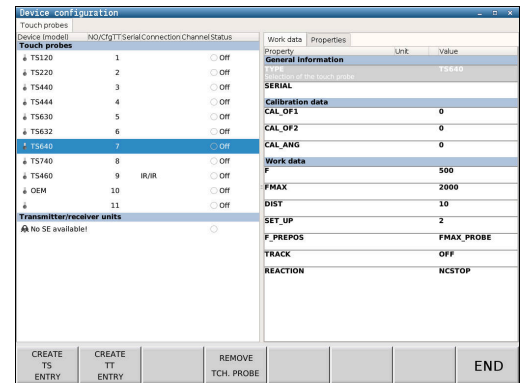
The **Set up touch probes** menu includes the following items:

■ **Touch probes**
■ **Transmitter/receiver units**
■ **Work data** tab
　■ **General information**
　■ **Calibration data**
　■ **Work data**
■ **Properties** tab
　■ **Connection settings**
　■ **Functions**
　■ **Current IR touch probe data**

Touch probes

Under **Touch probes**, the system displays all touch probes configured for the control.

The **Touch probes** menu item is structured as follows:

Contents	Explanation
Device (model)	Name of the touch probe
NO/CfgTT	Number in the touch probe table
Serial	Serial number of the touch probe
Connection	Connection type of the touch probe, e.g. **radio/IR**
Channel	Number of the radio channel
Status	Status of the touch probe, e.g. **Off**

Transmitter/receiver units

Under **Transmitter/receiver units**, the system displays all transceivers configured for the control.

Work data tab

On the **Work data** tab, you can configure default values for use with the selected touch probe.

The **Work data** tab contains the following menu items:

- **General information**
- **Calibration data**
- **Work data**

To open the **Work data** tab:

▶ Open the **Set up touch probes** menu

▶ Select the **Work data** tab

> The control opens the **Work data** tab.

General information

The **General information** menu item shows information on the currently selected touch probe:

Contents	Explanation
TYPE	Selected touch probe
SERIAL	Serial number of the selected touch probe

Calibration data

You will find the calibration data in your tool database.

Further information: "Touch probes", Page 607

Work data

The **Work data** menu item displays default values for the selected touch probe. When you program touch probe cycles, the control will use these default values.

Contents	Explanation
F	Feed rate for probing mode
FMAX	Rapid traverse in probing cycle
DIST	Maximum measuring range
SET_UP	Set-up clearance
F_PREPOS	Feed rate for pre-positioning
TRACK	Orientation of the touch probe
REACTION	Behavior in case of collision with the touch probe

Properties tab

The **Properties** tab contains further information on the status of the selected touch probe.

The **Properties** tab contains the following menu items:

- **Connection settings**
- **Functions**
- **Current IR touch probe data**

To open the **Properties** tab:

▶ Open the **Set up touch probes** menu

▶ Select the **Properties** tab

> The control opens the **Properties** tab.

Connection settings

Under **Connection settings**, you can select the type of triggering in case of deflection or when switching the touch probe on or off:

Connection setting			
Switching on/off	IR	Radio	Mechanical
Deflection	IR	Radio	Mechanical

Functions

Under **Functions**, you can select the transmitter unit to be triggered.
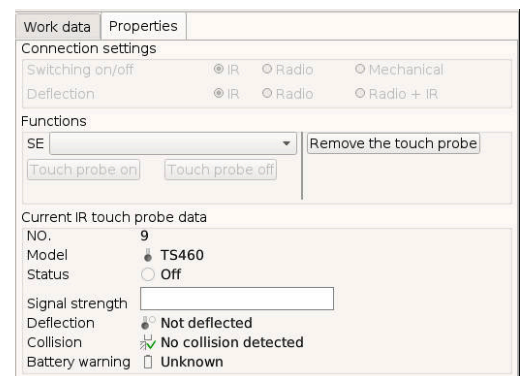
Current IR touch probe data

Select the **Current IR touch probe data** menu item to display an overview of the current status of the touch probe.

Under **Current IR touch probe data**, the following items are displayed:

Contents	Explanation
NO.	Number in the touch probe table
Model	Type of touch probe
Status	Touch probe active or inactive
Signal strength	Display of the signal strength in the bar chart. A full bar indicates an optimum connection.
Deflection	Stylus deflected or not deflected
Collision	Collision or no collision detected
Battery warning	Indicates the battery status; if the charge falls below the limit indicated by the vertical bar, the control displays a warning.

Creating a new touch probe

To create a new touch probe, proceed as follows:

▶ Open the **Set up touch probes** menu

CREATE
TS
ENTRY

    ▶ Press the **CREATE ENTRY** soft key

    > The control creates a new touch probe entry that will be displayed under **Touch probes**.

    ▶ Add the data for the new touch probe in the menu items explained above:

        ■ **General information**

        ■ **Calibration data**

        ■ **Work data**

    > The touch probe will be connected.

Removing a touch probe

To delete an existing touch probe, proceed as follows:

▶ Open the **Set up touch probes** menu
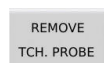
REMOVE
TCH. PROBE

    ▶ Press the **Remove the touch probe** soft key

    > The control opens the **Remove touch probe?** dialog.

OK

    ▶ Press the **OK** soft key

    > The touch probe will be deleted.

Suppressing touch probe monitoring

While you are using a touch probe, the **Touch probe not ready** error message might be displayed. This can have various causes.

These are the situations that may cause the **Touch probe not ready** error message to display:

- The touch probe is not connected
- The battery in the touch probe is dead
- No connection between the IR touch probe and the receiver unit

The error message immediately stops any machining operation and disables the manual axis keys. If you want to position the touch probe anyway, make sure to disable touch-probe monitoring.

NOTICE

Danger of collision!

The **TCH PROBE OFF** function suppresses the corresponding error message. In addition, the control does not perform an automatic collision check with the stylus. Because of this behavior, you must check whether the touch probe can retract safely. There is a risk of collision if you choose the wrong direction for retraction!

▶ Carefully move the axes in the **Machine** mode of operation

---

To suppress touch-probe monitoring, proceed as follows:

▶ Select **Setup**
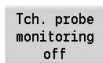
| Tch. probe monitoring off |

▶ Press the **TCH PROBE OFF** soft key

> The control disables touch-probe monitoring for 30 seconds.

> The control issues the error message **The touch probe monitor is deactivated for 30 seconds**.

> You can now position the touch probe within the working space during a period of 30 seconds.

Calibrating the tool touch probe

The **Touch probe calibration** function allows you to determine the exact position values of the tool touch probe.

Measuring the touch probe position:

▶ Select **Setup**

▶ Select **Touch probe**

▶ Select **Touch probe calibration**

▶ Pre-position the tool for the first measuring direction

▶ Set the positive or negative traverse direction
  +/-

▶ Press the soft key for this direction (e.g. −Z direction)
  -Z

▶ Press the **NC start** key
  > The tool moves in the measuring direction.
  > The position of the touch probe is measured and saved when the tool touch probe releases a trigger signal.
  > The tool returns to the starting point.

Back ▶ Press the **Back** soft key to terminate the calibration process
  > The calibration values measured are saved.
  ▶ Pre-position the tool for the next measuring direction and repeat the procedure (max. 4 measuring directions)
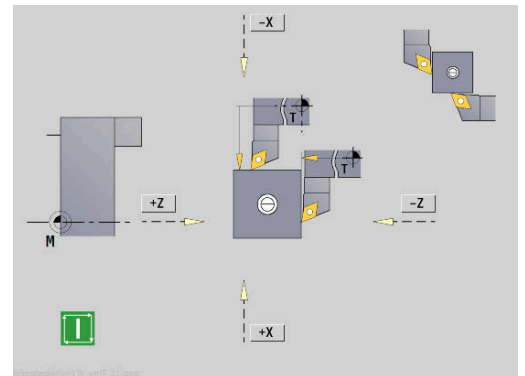
Displaying operating times

In the **Service** menu, you can view different operating times.

Operating time	Meaning
Control on	Operating time of the control since being put into service
Machine on	Operating time of the machine tool since being put into service
Program run	Duration of controlled operation since being put into service

> ⚙ Refer to your machine manual.
>
> Your machine tool builder can provide further operating time displays.

Displaying operating times:

▶ Select **Setup**

▶ Select **Service**

▶ Select **Display operating times**

Configuring the HR 550 FS wireless handwheel

Application

Select the **Set up wireless handwheel** menu item to configure the HR 550 FS wireless handwheel. The following functions are available:

- Assigning the handwheel to a specific handwheel holder
- Setting the transmission channel
- Analyzing the frequency spectrum for determining the optimum transmission channel
- Select transmitter power
- Statistical information on the transmission quality

To **Set up wireless handwheel**:

▶ Select **Setup**

▶ Select **Service**

▶ Select **Set up wireless handwheel**

Assigning the handwheel to a specific handwheel holder

▶ Make sure that the handwheel holder is connected to the control hardware.
▶ Place the wireless handwheel you want to assign to the handwheel holder in the handwheel holder
▶ Select the **Setup** menu item
▶ Select the **Service** menu item
▶ Select the **Set up wireless handwheel** menu item
▶ Click the **Connect HW** button
> The control saves serial number of the inserted wireless handwheel and shows it in the configuration window on the left next to the **Connect HW** button.
▶ Save the configuration and close the configuration menu: Press the **END** button
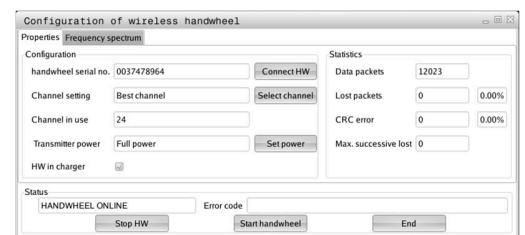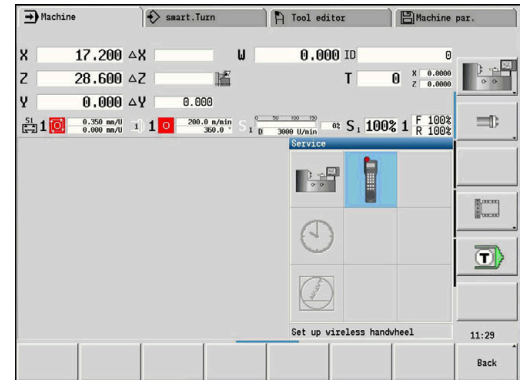
Setting the transmission channel

If the wireless handwheel is started automatically, then the control tries to select the transmission channel providing the best transmission signal. If you want to set the transmission channel manually, proceed as follows:

▶ Select the **Setup** menu item

▶ Select the **Service** menu item

▶ Select the **Set up wireless handwheel** menu item

▶ Click the **Frequency spectrum** tab

▶ Click the **Stop HW** button

> The control stops the connection to the wireless handwheel and determines the current frequency spectrum for all 16 available channels.

▶ Note the number of the channel with the least amount of radio traffic (smallest bar)

▶ Click the **Start handwheel** button to reactivate the radio handwheel

▶ Click the **Properties** tab

▶ Click the **Select channel** button

> The controls shows all available channel numbers

▶ Click the number of the channel that the control has found to have the least amount of radio traffic

▶ To save the configuration and exit the configuration menu, press the **END** button

Selecting the transmitter power

ⓘ   A reduction in transmission power decreases the range of the wireless handwheel.

▶ Select the **Setup** menu item

▶ Select the **Service** menu item

▶ Select the **Set up wireless handwheel** menu item

▶ Click the **Set power** button

> The control displays the three available power settings. Click the desired setting.

▶ To save the configuration and exit the configuration menu, press the **END** button
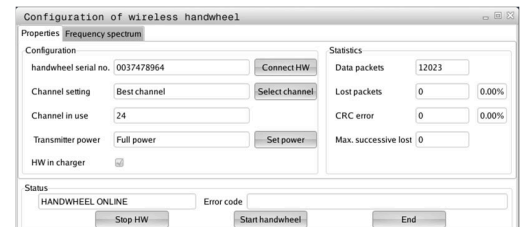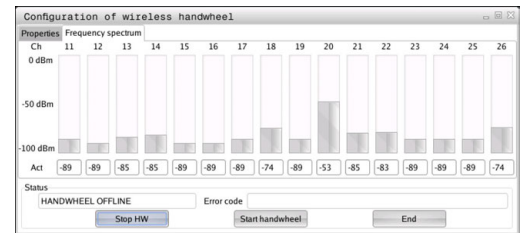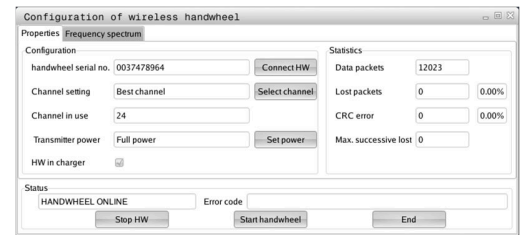
Statistical data

To display the statistical data, proceed as follows:

▶ Select the **Setup** menu item

▶ Select the **Service** menu item

▶ Select the **Set up wireless handwheel** menu item

> The control displays the configuration menu with the statistical data.

Under **Statistics**, the control displays information about the transmission quality.

If the received signal quality is impaired and no longer ensures a perfect, safe stop of the axes, the wireless handwheel will perform an emergency stop.

The displayed value **Max. successive lost** indicates whether reception quality is poor. If the control repeatedly displays values greater than 2 during normal operation of the wireless handwheel within the desired range of use, then there is a high risk of an undesired disconnection. This can be corrected by increasing the transmitter power or by changing to another channel with less radio traffic.

If this occurs, try to improve the transmission quality by selecting another channel or by increasing the transmitter power.

Further information: "Setting the transmission channel", Page 155

Further information: "Selecting the transmitter power", Page 155
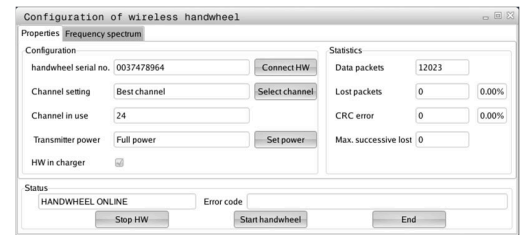
Setting the system time

The **Adjust the system time** function allows you to set the time on your control.

> ℹ️ You will need a mouse to navigate the **System time** input form.
>
> Use the **Month** and **Year** soft keys to increment or decrement the respective settings.
>
> To use an NTP server for setting the time, select a server from the server list first.

Setting the system time:

▶ Select **Setup**

▶ Select **Service**

▶ Select **Adjust the system time**

▶ Select **Synchronize the time over NTP server** (if available)

▶ Select **Set the time manually**

▶ Select **Date**

▶ Enter the **Time**

▶ Select **Time zone**

▶ Press the **OK** soft key

TNCdiag

> ℹ️ This function is available only to authorized specialists!
>
> Only use this function after consultation with your machine tool builder.

The **TNCdiag** function evaluates status and diagnostic information of HEIDENHAIN components with emphasis on the drive motors, and presents it graphically.

The **TNCdiag** application provides the following:

- Status and diagnostic information about the HEIDENHAIN components (drive electronics, encoders, input/output devices, etc.) connected to the control
- History of the recorded data
- Replacement of DriveDiag for Gen 3

> 📖 For more information, please refer to the **TNCdiag** documentation.

6.6 Tool measurement

The control supports tool calibration.

- By touch-off: The setup dimensions are determined by comparing a tool with an already measured tool
- By touch probe: Stationary or swiveling in the working space (installed by the machine tool builder)
- By optical gauge (installed by the machine tool builder)

Calibration by touch-off is always available. If a touch probe or an optical gauge is installed, select these measuring methods by soft key.

If the tool dimensions are already known, you can enter the setup dimensions directly in **Tool editor** mode of operation.

ⓘ
- The compensation values are deleted during tool measurement.
- Please note that for drilling and milling tools the center is measured.
- The tools' type and orientation determine how they are measured. Note the help graphics

---

Touch-off

You measure the dimensions relative to a calibrated tool by touching the tool off.

To find the tool dimensions by touch-off with the tool:

▶ In the tool table, enter the tool you want to measure

S ⬛/T ⬛ ⋙F	▶ Insert the reference tool and enter the tool number in the **TSF** dialog
	▶ Turn an end face and define this coordinate as the workpiece datum
S ⬛/T ⬛ ⋙F	▶ Return to the **TSF** dialog and insert the tool to be measured
Measure tool	▶ Press the **Measure tool** soft key
	▶ Touch the face with the tool
Transfer Z	▶ Enter the value **0** for **Measuring pt. coordinate Z** (workpiece datum) and save it
	▶ Turn the measuring diameter
Transfer X	▶ Enter the diameter value as **Measuring pt. coordinate X** and save it
Save radius	▶ For turning tools, enter the cutting edge radius and load it into the tool table
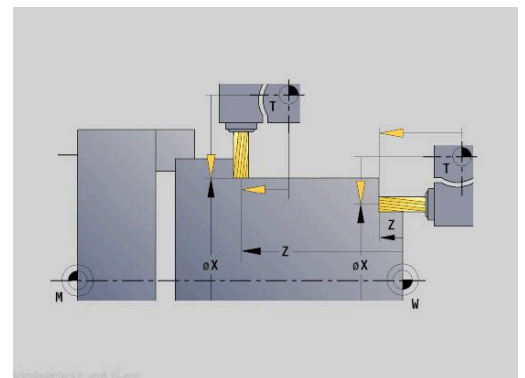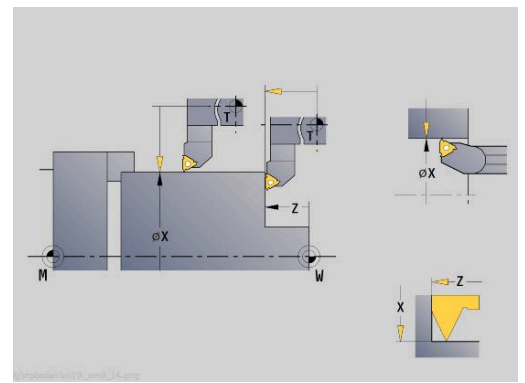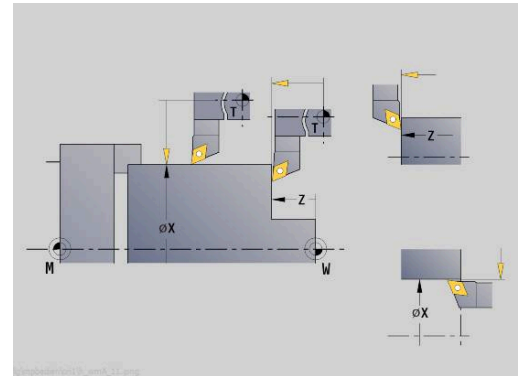
Touch probe (tool touch probe)

⚙️   Refer to your machine manual.
    Your machine manufacturer enables this function.

Finding the tool dimensions by using a touch probe:

▶   In the tool table, enter the tool you want to measure

S �'T
⬇ ⅲF

▶   Insert the tool and enter the tool number in the **TSF** dialog

| Measure
| tool

▶   Press the **Measure tool** soft key

| Measure
| probe

▶   Press the **Measure probe** soft key

▶   Pre-position the tool for the first measuring direction

| +/-

▶   Set the positive or negative traverse direction

| -Z

▶   Press the soft key for this direction (e.g. −Z direction)

🔲

▶   Press the **NC start** key
>   The tool moves in the measuring direction.
>   When it contacts the touch probe, the control calculates and saves the set-up dimensions.
>   The tool returns to the starting point.
▶   Pre-position the tool for the second measuring direction

| -X

▶   Press the soft key for this direction (e.g. −X direction)

🔲

▶   Press the **NC start** key
>   The tool moves in the measuring direction.
>   When it contacts the touch probe, the control calculates and saves the set-up dimensions.

| Save
| radius

▶   For turning tools, enter the cutting edge radius and load it into the tool table
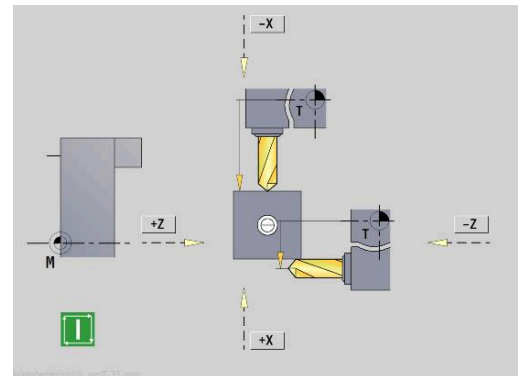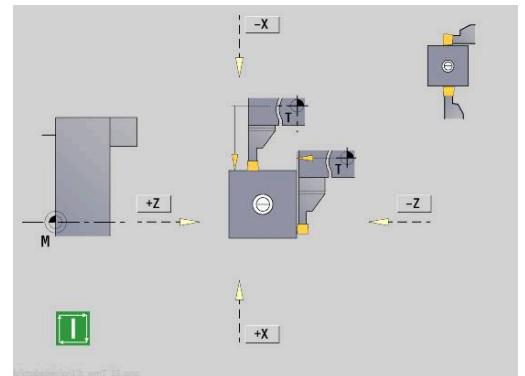
Optical gauge

> ⚙ Refer to your machine manual.
> Your machine manufacturer enables this function.

To find the tool dimensions by using an optical gauge:

▶ In the tool table, enter the tool you want to measure

[S ◪/T / ⬚ ⋀F]

    ▶ Insert the tool and enter the tool number in the **TSF** dialog

[Measure tool]

    ▶ Press the **Measure tool** soft key

[Measure optical]

    ▶ Press the **Measure optical** soft key

[Transfer Z]

    ▶ Position the tool at the cross hairs of the optical gauge by using the axis keys or the handwheel

    ▶ Save the tool dimension in Z

    ▶ Save the tool dimension in X

[Transfer X]

[Save radius]

    ▶ For turning tools, enter the cutting edge radius and load it into the tool table
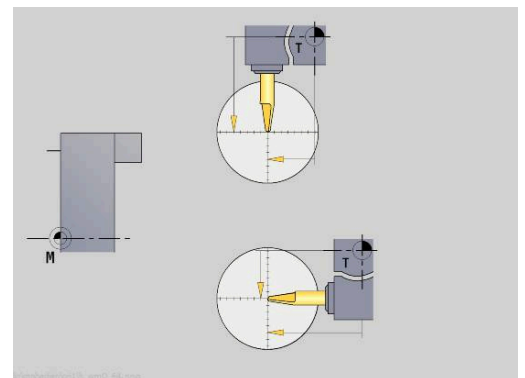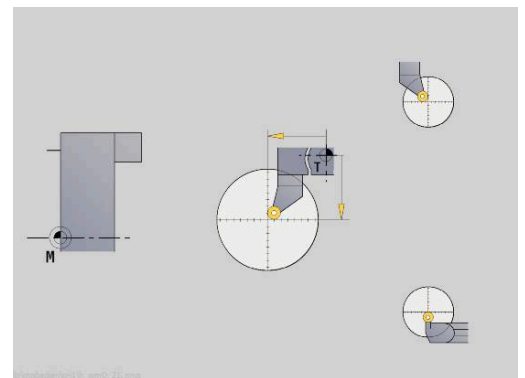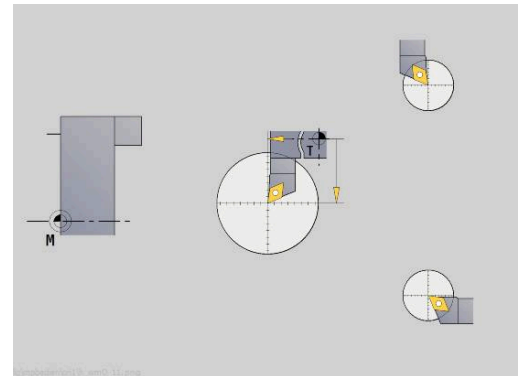
Tool compensation

The tool compensation in X and Z as well as the special compensation for recessing tools and button tools compensate for wear of the cutting edge.

> **i** A compensation value must not exceed +/−10 mm.

You can set tool compensations with the handwheel or enter them into a dialog field.

Setting tool compensation with the handwheel:

S ⬛T ⬛ ⌁F	▶	Select **Set T, S, F** (is only selectable in **Machine** mode of operation)
Tool correct.	▶	Press the **Tool correct.** soft key
Set by handwheel	▶	Press the **Set by handwheel** soft key if required
X compns. of tool	▶	Press the **X compns. of tool** (or **Z compns.**) soft key
	▶	Determine the compensation value with the handwheel
	>	The value will be displayed in the distance-to-go display.
Save	▶	Transfer the compensation value to the tool table
	>	The T display shows the new compensation value.
	>	The distance-to-go display is cleared.
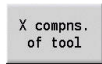
Defining tool compensation:
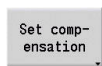
S ⬛T ⬛ ⌁F	▶	Select **Set T, S, F** (is only selectable in **Machine** mode of operation)
Tool correct.	▶	Press the **Tool correct.** soft key
Set comp- ensation	▶	Press the **Set compensation** soft key if required
Save	▶	Transfer the compensation value to the tool table
	>	The T display shows the new compensation value.
	>	The distance-to-go display is cleared.

Deleting tool compensation values:

▶ Select **Set T, S, F** (is only selectable in **Machine** mode of operation)
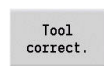
▶ Press the **Tool correct.** soft key

▶ Press the **Delete** soft key

▶ Delete compensation value in X (or Z)

6.7   Manual operation

When **machining a workpiece manually**, you move the axes with the handwheel or manual direction keys. You can also use Teach-in cycles for machining more complex contours (semi-automatic mode). The paths of traverse and the cycles, however, are **not stored**.

After switch-on and homing, the control is in **Machine** mode of operation. This mode remains active until you select the **Teach-in** submode or the **Program run** submode. "Machine" displayed in the header indicates that you are in **Manual mode**.

> ℹ️ Define the workpiece datum and enter the machine data before you start machining.

Tool change

Enter the tool number of the tool ID number in the **TSF** dialog. Check the tool parameters.

T0 does not define a tool. This also means that it does not contain any data on tool length, cutting radius, etc.

Spindle

Enter the spindle speed in the **TSF** dialog. To start and stop spindle rotation, press the spindle keys on the machine operating panel. The **Stopping angle A** in the **TSF** dialog makes the spindle always stop at this position.

> ⚙️ Refer to your machine manual.
>
> The maximum executable spindle speed varies depending on the machine. It may differ significantly from the maximum programmable spindle speed.
>
> The machine tool builder defines the maximum executable spindle speed in the machine parameters.

Handwheel operation

> ⚙️ **Further information:** Refer to your machine manual

Manual direction keys

With the manual direction keys, you can move the axes at the programmed feed rate or at rapid traverse. Enter the feed rate in the **TSF** dialog.

> ℹ️ ■ **Feed rate**
>   ■ If the **spindle is rotating**: Feed per revolution [mm/rev]
>   ■ If the **spindle is not rotating**: Feed per minute [m/min]
>   ■ Feed rate at **rapid traverse** : Feed per minute [m/min]

Teach-in cycles in Machine mode of operation

▶ Set the spindle speed

▶ Set the feed rate

▶ Insert the tool, define the tool number and check the tool data (**T0** is not permitted)

▶ Approach the cycle start point

▶ Select the cycle and enter the cycle parameters

▶ Graphically check the cycle run

▶ Run the cycle

> ℹ️ The entries last made in a cycle dialog remain in memory until a new cycle is selected.

6.8   Teach-in submode (option 8)

Teach-in submode

In **Teach-in** submode, you machine a workpiece step by step with the help of teach-in cycles. The control **memorizes** how the workpiece was machined and stores the machining steps in a cycle program, which you can call up again at any time. The **Teach-in** submode can be switched on with the **Teach-in** soft key; the header informs you that it is active.

Each Teach-in program is given a name and a short description. The individual cycles of a cycle program are listed as blocks and are numbered in ascending order. The block number has no meaning for the program run. The cycles are run after each other. When the cursor is located on a cycle block, the control displays the cycle parameters.

The cycle block contains:

- Block number
- Tool used (number and tool ID number)
- Cycle name
- Number of the ICP contour or of the DIN subprogram (after the **%** character)
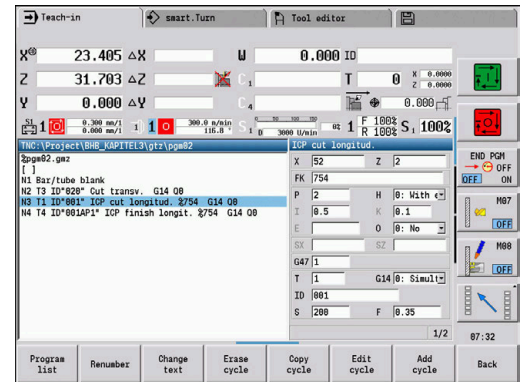
Programming Teach-in cycles

When creating a new Teach-in program, you program each cycle in the following sequence of actions **Enter – Simulate – Execute – Save**. The individual cycles form the cycle program.

You can change Teach-in programs by simply editing the necessary cycle parameters, and delete or add cycles as required.

> **i** The machine parameter **cycleAlwaysSave** (no. 604903) defines whether a cycle can be saved without previous simulation or execution.

When you exit the **Teach-in** submode or switch off the machine, the Teach-in program is kept. When you call an ICP cycle, the control displays a soft key for switching to the ICP contour editor.

You can program DIN subprograms in the smart.Turn editor and then integrate them into a DIN cycle. To open the smart.Turn editor, either use the **Edit DIN** soft key when selecting the DIN cycle, or use the operating-mode key.

Soft keys

Program list	Switch to the **select cycle programs** function
Renumber	Renumber the block numbers of the cycles
Change text	Enter / edit the program description
Erase cycle	Delete the selected cycle
Copy cycle	Copy the cycle parameters into a buffer memory Example: Copy the parameters of the roughing cycle to the finishing cycle
Insert	This soft key is available once you press the **Copy cycle** soft key. Insert copied data from the buffer
Edit cycle	Change the cycle parameter or cycle mode. The cycle type, however, cannot be changed.
Add cycle	Insert a new cycle below the highlighted block

6.9   Program run submode

Loading a program

In **Program run** submode, you use teach-in programs, DIN programs or automatic jobs for parts production. You cannot change the programs in this mode. The **Simulation** submode, however, allows you to check the programs before you run them. The control also offers the Single block and the Continuous run modes with which you can **machine step by step** the first workpiece of a whole batch.

The smart.Turn programs are saved as DIN programs (***.nc**). Automatic jobs (***.job**) are also generated in **smart.Turn** mode of operation.

By default, **Program run** submode automatically loads the most recently used program. Use the **autoPgmSelect** machine parameter (no. 601814) to specify that the program will not be selected automatically.

To load another program, proceed as follows:

Program list	▶ Press the **Program list** soft key
	> The control displays the teach-in programs.
	▶ Alternative: Display the **DIN** program
DIN	
	▶ Select the Teach-in program or DIN program
Open	▶ Press the **Open** soft key

You can start a Teach-in or smart.Turn program at any desired block to resume a machining operation after an interruption (Start blck search).

The **Program run** submode can be switched on by soft key; the header informs you that it is active.

Further information: "Program management", Page 188

ℹ Programs selected in **Program run** submode are protected from deletion.

To enable the file for deletion, exit the program block display by pressing the **Back** soft key.
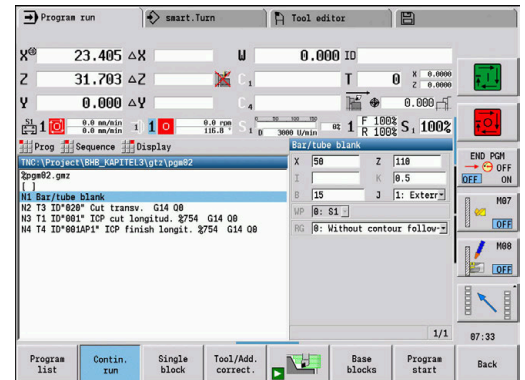
Comparing tool lists

While a program is being loaded, the control compares the current turret assignment list with the tool list of the program. If tools are used in the program that are not in the current turret list or are located in another pocket, an error message is displayed.

After the error message has been confirmed, the program-dependent tool list is shown for checking.

You can overwrite the current turret assignment list with the **Load tool** soft key. If you press the **Cancel** soft key, the program cannot be started. The tool list of the program must match the current turret assignment list.
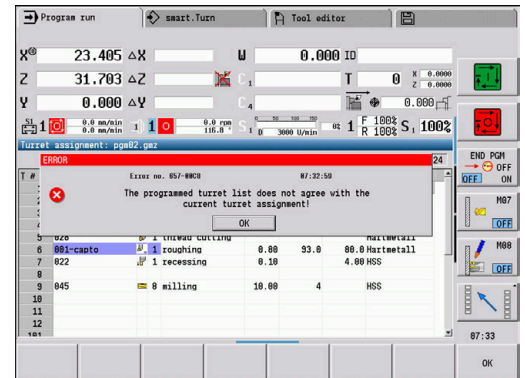
NOTICE

Danger of collision!

If you press the **Load tool** soft key, the current turret assignment list will be overwritten completely and irreversibly with the tool list of the program. The lists will then not be compared again for whether they match. There is a danger of collision during subsequent machining operations!

▶ Check the turret assignment list manually after it has been overwritten

---

> **i** This function is also available on machines with a tool magazine. The control uses the magazine list instead of the turret list.

Before executing a program

Faulty programs

The control checks the programs during loading up to the **MACHINING** section. If it detects an error (for example, an error in the contour description), it displays the error symbol in the header. You can then press the **Info** key for detailed information on the error. The machining section of a program, and therefore all traverse, is not interpreted until after **NC start**. If there is an error here, the machine stops with an error message.

> **i** Checking the NC programs in **Simulation** submode helps you to find programming errors or syntax errors in time before actual machining.

- **Checking the cycles and cycle parameters**: The control displays the Teach-in/DIN program in the list window. With Teach-in programs, the parameters of the cycle that is highlighted are displayed.
- **Graphic verification**: The program run can be verified in **Simulation** submode.

Further information: "Simulation submode", Page 544

Block scan

> ⚙ Refer to your machine manual.
>   This function must be set up by your machine
>   manufacturer.

> ⓘ Block scan (mid-program startup) is not available for
>   **Reference machining** operations required during **load
>   monitoring** (option 151).

> ⓘ Block scan (mid-program startup) is not available for
>   non-circular turning with a coupled spindle (option 135,
>   Synchronizing Funct.). Select an NC block before or after
>   the non-circular turning program section.

Mid-program startup means starting an NC program at a selected
point. In smart.Turn programs, you can start the program at any NC
block.

The control starts program run from the cursor position. The starting
position is not changed by a previous graphic simulation.

> ⓘ In the **execNextStartBlock** machine parameter (601810),
>   you can define whether program run after a block scan
>   will start at the selected NC block or at the subsequent NC
>   block.

NOTICE

Danger of collision!

The control brings the machine into the situation in which it would be in a normal program run just before the startup block (e.g. previous tool or previous tilting situation). There is a danger of collision during the tilting movement and the rotation!

▶ Pre-position the slide so that the tool carrier (e.g. turret) can tilt without collision

▶ Pre-position the slide so that the axes are able to move to the last programmed position before reentry without collision

HEIDENHAIN recommends starting at an NC block immediately after a **T** command.

The control restores the previous machine situation by proceeding in the following sequence:

- Insert the tool
- Position the axes in the configured or selected sequence
- Switch the spindle on

Soft key	Function
Search backwards	Search backward for soft key information (e.g., **Next T**). Backward search will remain enabled until you disable it.
Next UNIT	Go to the next **UNIT**.
Next T	Jump to the next tool call
Next L	Jump to the next subprogram call
Find text	Jump to the next occurrence of the text entered
Program start	Jump to the machining start at the beginning of the program

Program execution

The selected Teach-in or DIN program is executed as soon as you press **NC start**. You can interrupt machining at any time by pressing **NC stop**.

During program run, the cycle (or DIN block) that is currently being run is highlighted. With Teach-in programs, the parameters of the cycle currently being run are displayed in the input window.

You can influence the program run with the soft keys listed in this table.

Soft keys

Program list	Select the Teach-in program or smart.Turn program
Contin. run	Teach-in program: ■ **On**: Cycles are run continuously, one after the other, up to the next tool change to be acknowledged ■ **Off**: Stop after every cycle. Start the following cycle with **NC start**. smart.Turn program: ■ **On**: Program execution without any interruption ■ **Off**: Stop before **M01** command
Single block	■ **On**: Stop after every traverse path (basic block). Start the next path with **NC start** (Recommendation: Single block should be used together with the basic-block display) ■ **Off**: Run cycles and DIN commands without interruption
Tool/Add. correct.	Input of tool compensation values or additive compensation values **Further information:** "Entering compensation values during program run", Page 178
	Activate the **Simulation** submode
Base blocks	■ **On**: The traversing and switching commands are shown in **DIN format** (basic blocks) ■ **Off**: Teach-in or DIN program is displayed
Program start	In combination with DIN programs, the soft key is available once you press the **Start blck search** soft key. The cursor returns to the first block of the Teach-in program or DIN program
Start blck search	Press to enter into an NC program at a selected point **Further information:** "Block scan", Page 170

Moving the machine axes during a stop

While a program run is stopped, you can manually move the axes. When you then press **NC Start** to resume program run, the control will move the axes back to their original positions.

The machine manufacturer uses the optional machine parameter **CfgBackTrack** (122000) to define how the control returns to the previous position.

If the machine parameter is inactive, the control uses a predefined sequence.

If the machine parameter is active, the control remembers the path and returns on this path to the original position.

> **ℹ** If you interrupt program run during a tapping cycle, you can manually retract the tool from the hole in the Z axis. The control moves the spindle on a path matching the traverse.
>
> If the optional machine parameter **CfgBackTrack** (no. 122000) is active, use the **Start blck search** soft key to resume program run after the manual traverse.

No.pieces

> **⚙** Refer to your machine manual.
>
> This function must be enabled and adapted by the machine manufacturer.

You can predefine the number of pieces to be machined. The control will then run the program until the specified quantity is reached.

To define the value for No.pieces:

▶ Select the **Flow** menu item

▶ Select the **No.pieces** menu item
> The control opens the **No.pieces** form.

Cycle parameters:

- **MP**: **Default unit quantity**
- **P**: **Actual unit quantity**

With the **Delete qty. parts** soft key, you can reset the workpiece counter.

Skip level

Before you can set and activate skip levels, you need to define them in the program.

Further information: smart.Turn and DIN Programming User's Manual

To set a Skip level:

▶ Select the **Flow** menu item

▶ Select the **Skip level** menu item

> The control opens the **Set the skip level** form.

Cycle parameters:

- **NR**: **Skip levels**

If you enter the value **2** in the **NR** parameter and press the **Save** soft key, the control sets and activates the skip level 2 and refreshes the display field. Furthermore, the next time the program is run, the NC blocks defined by the set or active skip level will not be executed by the control.

Further information: "Machine data display", Page 123

ℹ You can set and activate more than one skip level at a time by entering a string of numerals in the **NR** parameter. The entry **159** sets/activates the skip levels 1, 5 and 9.

To deactivate the skip levels, program the **NR** parameter without a value and press Save.

When setting and activating skip levels during program run, please remember that the control will respond with a delay due to the block scan.

Variables

⚙ Refer to your machine manual.

This function must be enabled and adapted by the machine manufacturer.

In **Program run** submode, it is possible to display and edit the variables defined in the **PROGRAM HEAD** program section.

Further information: smart.Turn and DIN Programming User's Manual

To display or hide Variables:

▶ Select the **Flow** menu item

▶ Select the **Variables** menu item

▶ Select the **Output on** menu item
> The **Variables** pop-up window is displayed.
▶ Alternative: Select the **Output off** menu item
> The **Variables** pop-up window is hidden.

To change Variables:

▶ Select the **Flow** menu item

▶ Select the **Variables** menu item

▶ Select the **Change** menu item
> If the **Variables** pop-up window is not visible, it will be displayed now.
> You can now change the variables as desired.

ℹ Please note that you can only change variables while the program is not running (not started yet or stopped).

Automatic job

Automatic job

In **Program run** submode, the control can execute several main programs successively without you needing to reselect and start these programs. For this purpose, create a program list (automatic job) in **smart.Turn** operating mode that is then run in **Program run** submode.

Specify a quantity for each main program in the list that defines how often this program is carried out before the next NC program is started. With job selection, you can specify an NC program from which the job is to be processed.

If the automatic job was interrupted under the following conditions, the control will save the interrupted program and the number of pieces already machined:

- **NC stop**
- Emergency stop
- Power failure

ⓘ   Programming notes:
- Automatic jobs (*.job) can only be created in the default directory. The NC programs used for the job, however, can be stored in any project directories.
- If the program list should be processed without intervention, the listed main programs must be exited with **M99**.
- **M30** stops the automatic job. With the **NC start** key, you can resume the automatic job.

To select a job:

▶ Select the **Prog** menu item

▶ Select the **Job selection** menu item

▶ Select **Automatic job**
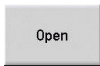
Open

▶ Press the **Open** soft key

↓

▶ Select the start program with the cursor if necessary
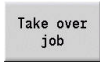
Take over job

▶ Confirm with the **Take over job** soft key

To resume a job after an interruption:

▶ Select the interrupted job

Interrupt program

▶ Press the **Interrupt program** soft key

> The control marks the interrupted program.

> The control sets the workpiece counter to the number of pieces already machined.

▶ Press the **NC start** key

To change the job display:

▶ The desired job must have been selected

▶ The cursor is located at the selected program to be started

▶ Select the **Display** menu item

▶ Select the **Switch off job list** menu item

> This menu item switches to the NC program display.

▶ Alternative: Select the **Switch on job list** menu item

> This menu item switches to the job display.

Entering compensation values during program run

Tool compensation

Entering tool compensation:

| Tool/Add. correct. | ▶ Press the **Tool/Add. correct.** soft key |

| Tool correct. | ▶ Press the **Tool correct.** soft key |

▶ Enter the tool number or select a tool from the tool list
▶ Enter the compensation values

| Save | ▶ Press the **Save** soft key |

> The valid compensation values are displayed in the input window and adopted.
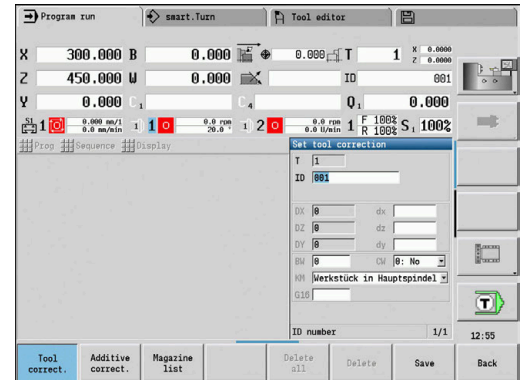
ℹ Programming notes:
- The entered values are added to the existing compensation values and are immediately effective in the display. The control moves in the compensation direction by the compensation value in the following traverse block.
- To delete a compensation, enter the current compensation value with the opposite algebraic sign.

Displaying tools from the selected NC program

To directly compensate the tools of the active NC program on machines with a magazine, the magazine list has an appropriate filtering function.

To filter the magazine list, proceed as follows:

| 🔲 | ▶ Select the **Machine** operating mode |

| Program run | ▶ Press the **Program run** soft key |

▶ Open the desired NC program

| Tool/Add. correct. | ▶ Press the **Tool/Add. correct.** soft key |

| Magazine list | ▶ Press the **Magazine list** soft key |

> The control opens the magazine list.

| Tools in program | ▶ Press the **Tools in program** soft key |

> The control shows the tools from the NC program currently opened.

ℹ If you have loaded an NC program, you can also use the **Tools in program** filter during the **Tool change** function.
Further information: "Editing the magazine list", Page 135

Additive compensation

The control manages 16 additive compensation values. You can edit the compensation values in **Program run** submode and activate them with **G149** in a smart.Turn program or in ICP finishing cycles.

Entering additive compensation:

▶ Press the **Tool/Add. correct.** soft key

[Tool/Add. correct.]

▶ Press the **Additive correct.** soft key

[Additive correct.]

▶ Enter the number of the additive compensation
▶ Enter the compensation values

[Save]

▶ Press the **Save** soft key
> The valid compensation values are displayed in the input window and adopted

Reading additive compensation:

[Tool/Add. correct.]

▶ Press the **Tool/Add. correct.** soft key

[Additive correct.]

▶ Press the **Additive correct.** soft key

▶ Enter the number of the additive compensation
▶ Place the cursor in the next input field
> The control displays the valid compensation values

[↓]

Deleting additive compensation:

[Tool/Add. correct.]

▶ Press the **Tool/Add. correct.** soft key

[Additive correct.]

▶ Press the **Additive correct.** soft key

▶ Enter the number of the additive compensation
▶ Press the **Delete** soft key
> Press the Delete soft key—the values of this compensation are deleted
▶ Alternative: Press the **Delete all** soft key
> All compensation values will be deleted
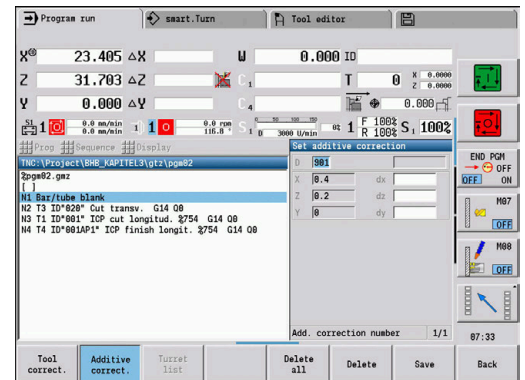
[Delete]

[Delete all]

ⓘ   ■ Entered values are added to the existing compensation values and are immediately effective in the display. The control moves in the compensation direction by the compensation value in the following traverse block
   ■ The compensation values are saved in an internal table and are available in any program
   ■ Delete all additive compensation values when you set up the machine again

---

Program execution in dry run mode

The dry run mode is used for fast program execution up to a point at which machining is to resume.

The following requirements must be met before starting a dry run:

- The control must be prepared by the machine tool builder for dry run. (The function is usually activated with a keylock switch or a key)
- The **Program run** submode must be activated

When dry run is activated, the spindle status and the spindle speed are **frozen**.

NOTICE

Danger of collision!

In dry run mode, the control performs all movements (except thread cuts) at rapid traverse. There is an increased danger of collision during dry run mode!

▶ Use the dry run mode only for **air cuts**
▶ Reduce the feed rate with the override potentiometer if necessary

---

After deactivation of dry run mode, the control returns to the programmed feed rates and the programmed spindle speed.

6.10 Load monitoring (option 151)

> ⚙ Refer to your machine manual.
>
> This function must be set up by your machine manufacturer.

> ℹ The following steps are required before you can use the load monitoring feature in **Program run** submode:
>
> ■ Define the respective machine parameters in the System section
> **Further information:** "List of machine parameters", Page 620
>
> ■ In **smart.Turn** operating mode, define the type of load monitoring with **G996** and the monitoring zone with **G995** in your program
> **Further information:** smart.Turn and DIN Programming User's Manual

During a machining operation with active load monitoring, the control compares the current utilization of the drives specified by **G995** with the respective limit values. To calculate the limit values for a monitoring zone defined with **G995**, the control uses reference values determined during reference machining as well as predefined factors specified in the machine parameters.

If the utilization limit 1 or the utilization sum limit is exceeded, the control issues a warning and sets the corresponding diagnostic bits in **Tool editor** mode of operation in order to identify the active tool as **worn-out**.

If the utilization limit 2 is exceeded, the control issues an error message, stops the machining operation and sets the corresponding diagnostic bits in **Tool editor** mode of operation in order to identify the active tool as **broken**.

You can manage the diagnostic bits in **Tool editor** mode of operation.

Further information: "Diagnostic bits", Page 581

> ℹ If tool life monitoring is active, the control will automatically insert a predefined replacement tool during the next tool call in order to replace the tool that has been flagged as **worn-out** or **broken**. As an alternative to the automatic evaluation of the diagnostic bits by the tool life monitoring function, you can also evaluate the diagnostic bits in your program.

i Keep in mind that load monitoring is not possible for hanging axes without counterweight!

i Note that load monitoring only works to a limited extent if the load changes are small. Therefore, monitor the drives that are subjected to significant loads, such as the main spindle.

i When face turning with a constant surface speed, remember that the load monitoring feature will monitor the spindle up to a maximum of 15 % of the nominal acceleration defined in the machine parameters. Since acceleration increases as a result of the change in rotational speed, the control only monitors the period after the first cut!

i The load monitoring function compares current utilization values with maximum limit values. For proper comparison, the utilization values must not be too low.

Since the utilization depends on the cutting conditions, the following example is provided for your guidance. It illustrates the required values for the machining of steel:

- **Longitudinal turning**: Cutting depth > 1 mm
- **Recessing**: Cutting depth > 1 mm
- **Drilling into solid material**: Hole diameter > 10 mm

Reference machining

During reference machining, the control determines the maximum utilization and the total utilization (utilization sum) for each monitoring zone. The determined values are used as reference values. To calculate the limit values for a monitoring zone, the control uses the determined reference values and the predefined factors specified in the machine parameters.

i During reference machining, the block scan feature is not available.

i Perform reference machining under the conditions in which the parts will later be produced, for example, with respect to feed rates, spindle speeds, and the type and quality of the tools you will be using.

Reference machining:

Program run	▶ Select the **Program run** submode and open the NC program
	▶ Activate load monitoring: Select the **Sequence** menu item
	▶ Select the **Load monitoring ON** menu item
	▶ Select reference machining: **Sequence** menu
	▶ Select the **Reference machining** menu item > The control shows the title bar with a green background.
	▶ Start reference machining: Press the **NC start** key > The control executes the machining operation and saves the reference data to a separate file. > After the successful completion of reference machining, the control displays an information message.
OK	▶ Press the **OK** soft key

> **ℹ** Program **M30** or **M99** to conclude reference machining. If the program is aborted during machining, no reference data is saved. In this case you need to execute reference machining again.

> **ℹ** Reference machining must be repeated if you make any changes to your program; for example:
> - Defining new zones
> - Deleting existing zones
> - Changing zone numbers
> - Changing, adding or removing axes within a zone
> - Changing feed rates or spindle speeds
> - Changing tools
> - Changing cutting depth values

Checking the reference values

After the successful completion of reference machining, check the determined reference values.

> **i** The load monitoring function compares current utilization values with limit values. For proper comparison, the reference values for utilization must not be too low. Check the determined values and, if required, remove monitored axes with a utilization rate of less than 5 % from the monitoring zone.

Meanings of the values:

- **Utilization**: Determined drive torque relative to the rated torque of the motor in **[%]**
- **Utilization sum**: Sum of the utilization values in the monitoring zone in **[%*ms]**

To open the reference values:

> ▶ Select the **Display** menu item

> ▶ Select the **Edit load data** menu item
> > The control opens the **Set load data** form containing the parameters listed below. In addition, the determined values are displayed as a bar graph

Cycle parameters:

- **ZO**: **Zone number** – Number of the monitoring zone
- **AX**: **Axis name** – Monitored axis
- **CH**: **Channel no.** – Selected channel
- **T**: **Tool pocket** of the active tool in the monitoring zone
- **ID**: **ID no.** – Name of the active tool in the monitoring zone
- **P**: **Utilization** – Maximum utilization during reference machining
- **PA**: **Utilization** – Maximum utilization during the current machining operation
- **PG1**: **Limit value** – Utilization limit 1
- **PG2**: **Limit value** – Utilization limit 2
- **W**: **Utilization sum** during reference machining
- **WA**: **Utilization sum** during the current machining operation
- **WGF**: **Limit value factor** – Factor for the limit of the utilization sum
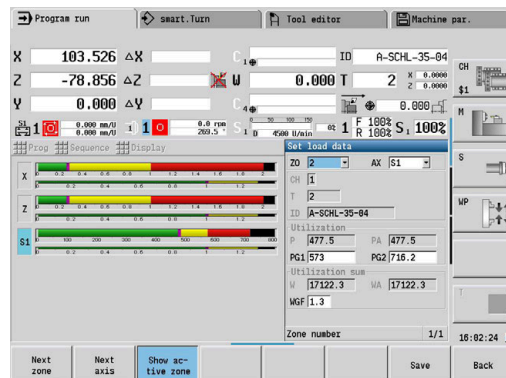
Bar graph:
- Thick upper bar (display in %):
  - **Green**: Range up to maximum **Utilization P** during reference machining.
  - **Yellow**: Range up to utilization limit 1 (**PG1**).
  - **Red**: Range up to utilization limit 2 (**PG2**).
  - **Magenta**: Maximum utilization during last machining operation (**PA**).
- Thin lower bar (display standardized to reference value 1):
  - **Green**: Range up to maximum utilization sum during reference machining (**W**).
  - **Yellow**: Range up to limit of the utilization sum (**WGF**).
  - **Magenta**: Maximum utilization sum during last machining operation (**WA**).

ℹ After reference machining, the value **W** equals **WA** and the value **P** equals **PA**. They are used as the reference values for calculating the limit values.

Adapting the limit values

After successful reference machining, the control uses the reference values and the predefined factors specified in the machine parameters to calculate the limit values.

You can adapt the calculated limit values as required for the subsequent production.

To adjust the limit values:

▶ Display the limit values: Select the **Display** menu item

▶ Select the **Edit load data** menu item

> The control opens the **Set load data** form

▶ Check the limit values

▶ If required, adapt the parameters **PG1**, **PG2** or **WGF**
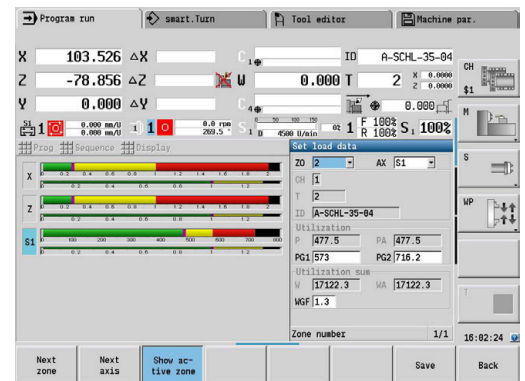
ℹ Be sure to adapt the correct limit values. First, press the **Next zone** and **Next axis** soft keys to select the form containing the limit values you want to change! As an alternative, you can use the selection lists of the **ZO** and **AX** parameters to select the correct form. Save the changes separately for each axis by pressing the **Save** soft key!

ℹ Reference machining does not have to be repeated after adapting the limit values. You can continue production with the adapted limit values.

Using load monitoring during production

ℹ️ Keep in mind that you cannot adapt the limit values during machining. Adapt the limit values before starting a machining operation.

During **Program run** submode, the control monitors the utilization and the utilization sum in each interpolator cycle. In parallel with machining, you can display a graph of the current utilization values for all monitored axes of the active zone.

To open the graph during machining:

Displaying the utilization values:

▶ Select the **Display** menu item

▶ Select the **Edit load data** menu item

> The control opens the **Set load data** form and additionally displays the determined values as a bar graph

Displaying the current utilization values:

```
Show ac-
tive zone
```

▶ Press the **Show active zone** soft key

> The control automatically switches to the current monitoring zone and displays the current utilization values as a bar graph
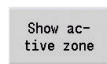
Bar graph:

- Thick upper bar (display in %):
    - **Green**: Current utilization (**PA**).
- Thin lower bar (display standardized to reference value 1):
    - **Green**: Current peak value between 0 and limit value 1 (**P**).
    - **Yellow**: Current peak value between P and limit value 1 (**PG1**).
    - **Red**: Current peak value between PG1 and limit value 2 (**PG1**).
- Thin lower bar (display standardized to reference value 1):
    - **Green**: Current utilization sum (**WA**).
    - **Yellow**: Current utilization sum up to the limit value (**WGF**).
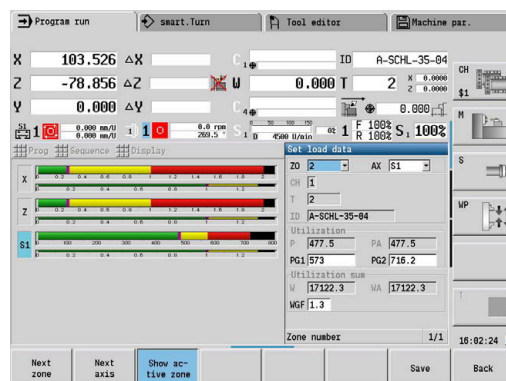
6.11 Graphic simulation

The **Simulation** submode enables you to check the machining sequence, the proportioning of cuts and the finished contour before actual machining.

In **Machine** operating mode and in **Teach-in** submode, this function simulates the execution of a single teach-in cycle; in **Program Run** submode, it simulates a complete teach-in program or DIN program.

A programmed workpiece blank is displayed in **Simulation** submode. The control also simulates machining operations that are executed with a traversable spindle or the C axis on the face or lateral surface. This allows you to check the complete machining process.

In **Machine** operating mode and **Teach-in** submode, the teach-in cycle you are currently processing is simulated. In **Program run** submode, the simulation starts from the cursor position. smart.Turn and DIN programs are simulated from the beginning of the program.

Further information: "Simulation submode", Page 544
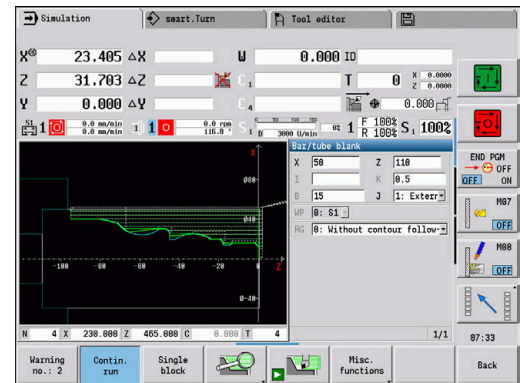
6.12 Program management

Program selection

By default, **Program run** submode automatically loads the most recently used program. Use machine parameter 601814 to specify that the program will not be selected automatically.

In the program selection, the programs available in the control are listed. Select the desired program or use the **ENT** key to go to the **File name** input field. In this input field, you limit the selection or enter the program name directly.

Program list	▶ Press the **Program list** soft key: Use the soft keys for selecting and sorting the programs

Soft keys in the program selection dialog

Details	Display the file attributes: ■ Size ■ Date ■ Time
DIN	Switch between Teach-in and DIN/smart.Turn programs
File manager	Open the file management soft-key menu **Further information:** "File manager", Page 190
Sorting	Open the sorting functions soft-key menu
Project	Open the project management soft-key menu **Further information:** "Project management", Page 191
Alphabetic keyboard	Open the alphabetic keyboard **Further information:** "Alphabetic keyboard", Page 78
Open	Open the program for the automatic start
Cancel	Close the program selection dialog. The program previously active in **Program run** submode remains effective
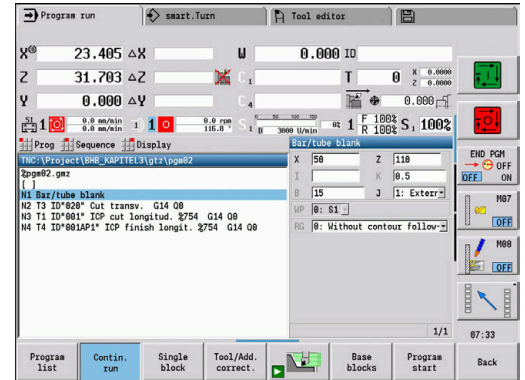
Soft keys for sorting functions

Details	Display the file attributes: ■ Size ■ Date ■ Time
Sort by file name	Sort the programs by file name
Sort by size	Sort the programs by file size
Sort by date	Sort the programs by change date
Update	Update the marked program
Reverse sorting	Reverse the sorting direction
Open	Open the program for the automatic start
Back	Return to program selection dialog

File manager

With the file management functions, you can copy, delete and otherwise manipulate files. You can select the program type (teach-in, smart.Turn or DIN program) before calling the program organization.

File manager soft keys

Paths / files	Switch between directory and file window
Cut out	Cut marked file
Copy	Copy marked file
Insert	Insert the file stored in the buffer memory
Rename	Rename marked file
Delete	Delete the marked file following confirmation prompt; program block display must not be open in any operating mode
Back	Return to the program selection dialog

Other soft keys

Details	Display details
Mark all	Mark all files
Update	Update the marked program
Write protection	Activate/deactivate write protection for the marked program
Alphabetic keyboard	Open the alphabetic keyboard
Back	Return to the program selection dialog

Project management

You can make your own project folder in the project management so that you can centrally manage associated files. When you create a project, a new folder is set up with the corresponding subfolder structure in the **TNC:\Project** directory. You can save your programs, contours and drawings to the subfolders.

The **Project** soft key activates the project management. The control shows you all existing projects in a tree structure. In project management, the control also opens a soft-key menu that allows you to create, select, and manage projects. To reselect the standard directory of the control, select the **TNC:\nc_prog** folder and press the **Select std. directory** soft key.

Soft keys project

New project	Create new project
Copy project	Copy marked project
Delete project	Delete marked project following confirmation prompt
Rename project	Rename marked project
Selection of standard	Open standard directory
Selection of PLC:	Open machine tool builder programs
Select project	Select marked project
Select std. directory	Select standard directory

ℹ️ You can select any project names. The subfolders (**dxf**, **gti**, **gtz**, **ncps** and **Pictures**) have predefined names that must not be changed.

All existing project folders are displayed in the project management. Use the file manager to switch between subfolders.

6.13 Conversion into DIN format

The "Convert to DIN" function enables you to convert a Teach-in program to a smart.Turn program with the same functionality. You can then optimize, expand such a smart.Turn program, etc.

Making a conversion

Converting into DIN format:

Cyc.prog. -> DIN	▶ Press the **Cycle program --> DIN** soft key (**main menu**)
	▶ Select the program to be converted
Cyc.prog. -> DIN	▶ Press the **Cycle program --> DIN** soft key (**program selection menu**)

The generated DIN program has the same program name as the Teach-in program.

Should the control encounter any errors during conversion, it generates an error message and cancels conversion.

If a program with the name used is open in the smart.Turn editor, the conversion must be confirmed with the **Overwrite** soft key. The control overwrites the program that is open in the smart.Turn editor.

6.14 Units of measure

The control is operating in either the **metric** or **inch** system. Depending on the unit of measure, the units and decimal places listed in the tables are used in the display values and entries.

Units

	Metric	Inches
Coordinates, lengths, path data	mm	inch
Feed rate	mm/rev or mm/min	in./rev or in./min
Cutting speed	m/min	ft/min (feet/min)

Number of decimal places in display values and entries

	Metric	Inches
Coordinate data and path data	3	4
Compensation values	3	5

The inch/metric setting is also evaluated for the display values and entries in the tool management.

Use the **unitOfMeasure** machine parameter (101101) to set the unit of measure to metric/inches. Changed metric/inch settings become effective immediately, with no need for a restart.

The basic block display also switches to inches.

> ℹ️ ■ The unit is defined in all NC programs; metric programs can be run while inch mode is active and vice versa
> ■ New programs are created in the selected unit
> ■ Refer to your machine manual if you want to know whether and how the handwheel resolution can be set to inches
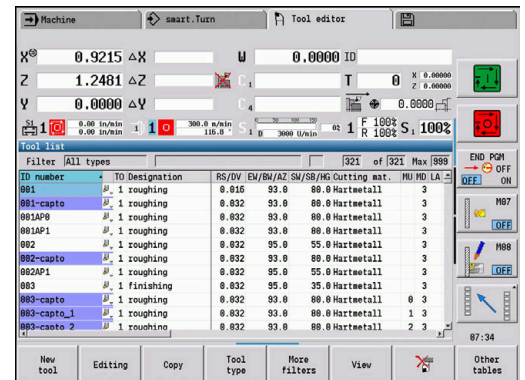
7

Teach-in

7.1   Working with cycles

Before you can use the cycles, you must set the workpiece datum
and ensure that the tools you are going to use have been described.
You enter the machine data (tool, feed rate, spindle speed) in **Teach-
in** submode together with the other cycle parameters. In **Machine**
mode, you must program this machine data before calling a cycle.

> **i**   The cutting data can be taken from the technology
> database using the **Proposed technology** soft key. For
> this database access, a machining type is permanently
> assigned to every cycle.

Define the individual cycles as follows:
- Position the tool tip with the handwheel or the jog buttons at the
  starting point of the cycle (only in **Machine** mode)
- Select and program a cycle
- Graphically test the cycle
- Execute the cycle
- Save the cycle (only in **Teach-in** submode)

Cycle starting point

In **Machine** mode, cycles will be executed from the current tool
position.

In **Teach-in** submode, you enter the starting point as one of the
parameters. The control moves to this position at rapid traverse
using the shortest path (diagonal) before executing the cycle.

NOTICE

Danger of collision!

The control does not automatically check whether collisions can
occur between the tool and the workpiece. There is danger of
collision during the approach movement!

▶ If necessary, program an additional rapid traverse path to a
   safe auxiliary position

Help graphics

The functions and parameters of the Teach-in cycles are illustrated in the graphic support window. These graphics usually show an external machining operation.

⟳	▶ The triple-arrow key allows you to switch between the help graphics for internal and external machining

Elements used in the graphic support window:

- Broken line: Rapid traverse path
- Continuous line: Feed path
- Dimension line with arrow head on one side: Directional dimension—the algebraic sign defines the direction
- Dimension line with arrow head on both sides: Absolute dimension—the algebraic sign has no effect

DIN macros

DIN macros (DIN cycles) are DIN subprograms.

Further information: "DIN cycles", Page 433

You can integrate DIN macros in Teach-in programs. Make sure that the DIN macros do not contain any datum shifts.

NOTICE

Danger of collision!

In **Teach-in** submode, all datum shifts contained in a DIN cycle (DIN macro) will be reset after the cycle has been executed. Danger of collision during subsequent machining operations!

▶ Use DIN cycles without datum shifts

Graphical test run (simulation)

Before executing a cycle, you can graphically test the contour details and the machining sequence.

Further information: "Simulation submode", Page 544

Contour follow-up in Teach-in submode

The contour follow-up function updates the originally defined workpiece blank with every machining step. The turning cycles take the current contour of the workpiece blank into account for the calculation of infeed and machining paths. Air cuts are avoided and approach paths optimized.

To activate contour follow-up in **Teach-in** submode, program a workpiece blank and select "With contour follow-up" in the **RG** parameter.

Further information: "Workpiece blank cycles", Page 204

> ℹ️ When contour follow-up is active, you can also use modal functions, such as interrupted feed or datum shift.
>
> Contour follow-up can only be used for turning operations and centric drilling.

Cycle run with active contour follow-up (**RG**: 1):

- Pressing the **NC start** key initiates a search for the selected cycle for mid-program startup
- The next **NC start** executes the **M** commands (e.g. direction of rotation)
- The next **NC start** positions the tool to the last programmed coordinates (e.g. tool change point)
- The next **NC start** runs the selected cycle

Cycle keys

You can execute a programmed teach-in cycle by pressing the **NC start** key. **NC stop** interrupts a running cycle. If you press **NC stop** during thread cutting, the tool is lifted off the contour and then stopped. The cycle has to be restarted.

During a cycle interruption you can:

- Resume cycle execution with the **NC start** key. The control will always resume execution of the cycle at the point of interruption, even if you have moved the axes in the meantime
- Move the axes with the manual direction keys or the handwheel
- Terminate the machining process with the **BACK** soft key

Switching functions (M functions)

The control generates all switching functions that are necessary for running a cycle.

The direction of spindle rotation must be defined in the tool parameters. Using the tool parameters, the cycles generate spindle trigger functions (**M3** or **M4**).

> ⚙️ Refer to your machine manual.
>
> On your lathe, other or additional M commands may apply to the automatically triggered switching functions.

Comments

You can assign a comment to an existing Teach-in cycle. The comment is inserted in brackets **[...]** below the cycle.

To add or edit comments:

▶ Create and select a cycle

Change text
   ▶ Press the **Change text** soft key

GOTO
   ▶ Press the **GOTO** key to show the alphabetic keyboard

   ▶ Enter the comment with the on-screen alphabetic keyboard

Save
   ▶ Apply the comment

Cycle menu

The main menu shows the cycle groups. Once a group has been selected, the menu items for the individual cycles appear.

You can use ICP cycles for complex contours, and DIN macros for technologically sophisticated machining operations. In cycle programs, the names of the ICP contours and DIN macros are at the end of the line of the cycle.

Some cycles offer optional parameters. That means, specific contour elements will only be machined if you set the corresponding parameters. The identification letters for optional parameters and parameters that are preassigned default values are displayed in gray.

The following parameters are used only in **Teach-in** submode:

- Start point **X**, **Z**
- Machine data **S**, **F**, **T** and **ID**
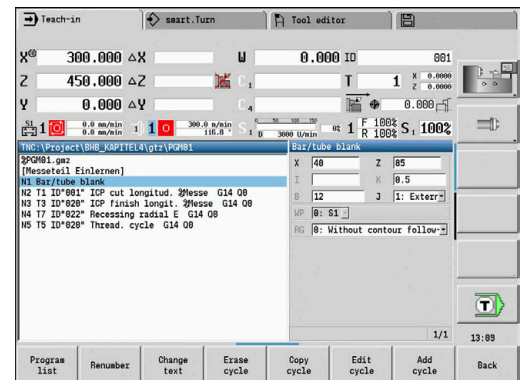
Menu item	Cycle groups
	Define workpiece blank
	Defining standard workpiece blanks or workpiece blanks with ICP
	Single cuts
	Positioning in rapid traverse, linear and circular single cuts, chamfers, and rounding arcs
	Roughing cycles lon/trans
	Roughing and finishing cycles for longitudinal and transverse machining
	Recessing cycle
	Cycles for recessing, contour recessing, undercuts and parting
	Thread cutting
	Thread cycles, relief turns and thread repair
	Drilling
	Drilling cycles and patterns for face and lateral-surface machining
	Millg
	Milling cycles and patterns for face and lateral-surface machining
	DIN cycle
	Including DIN macros

Soft keys in cycle programming: Depending on the type of cycle, you define the variants of the cycle by soft key.

Soft keys in cycle programming

Edit ICP	Call the interactive contour input
T-Change approach	Move to the tool change position
Spindle Stop M19	Activate spindle positioning (**M19**)
With return	■ **On:** Tool returns to starting point ■ **Off:** Tool remains at cycle end position
Finishing run	Switch to the finishing operation
Expanded	Switch to the expanded mode
Tool List	Open the turret list and tool list. You can load the tool from the list.
Take over position	Take over the actual X and Z positions in **Teach-in** submode
Proposed technology	Accept the proposed values for feed rate and cutting velocity from the database
Constant speed	■ **On:** Constant speed (rpm) ■ **Off:** Constant surface speed (m/min)
Pattern linear	Linear drilling and milling patterns on face or lateral surface
Pattern circular	Circular drilling and milling patterns on face or lateral surface
Input finished	Apply entered or changed values
Back	Cancel the current dialog

If you have confirmed the cycle with the **Input finished** soft key, a further soft-key row is displayed.

Soft keys in cycle programming

`Tool correct.`	Tool compensation **Further information:** "Tool compensation in Teach-in submode", Page 202
`Single block`	Initiate cycle in Program Run, Single Block
`Base blocks`	Display basic blocks
(simulation icon)	Display simulation
`Save`	Save the cycle (only with **Add cycle**)
`Over- write`	Overwrite the cycle (only with **Edit cycle**)
`Back`	Return to cycle description

> ℹ️ The machine parameter **cycleAlwaysSave** (no. 604903) defines whether a cycle can be saved without previous simulation or execution.

Tool compensation in Teach-in submode

Tool compensation in Teach-in submode

You can set tool compensations with the handwheel or enter them into a dialog field.

To define tool compensation:

▶ Define the cycle

`Input finished`	▶ Press the **Input finished** soft key
`Tool correct.`	▶ Press the **Tool correct.** soft key
`Set by handwheel`	▶ Press the **Set by handwheel** soft key
`Set comp- ensation`	▶ Alternative: Press the **Set compensation** soft key
`X compns. of tool`	▶ Press the **X compns. of tool** (or Z compns.) soft key
	▶ Determine the compensation value by handwheel; the value is shown in the distance-to-go display
	▶ Alternative: Enter the compensation value **dx** (or **dz**, **dy**)
`Save`	▶ Press the **Save** or **Overwrite** soft key

Addresses used in many cycles

Safety clearance G47

Safety clearances are used for approach and departure paths. If the cycle run takes a safety clearance into account during execution you will find the address **G47** in the dialog.

Proposed value:

Further information: "List of machine parameters", Page 620

Safety clearances SCI and SCK

The safety clearances **SCI** and **SCK** are considered for approach and departure paths in drilling and milling cycles.

- **SCI**: Safety clearance in the working plane
- **SCK**: Safety clearance in infeed direction

Proposed value:

Further information: "List of machine parameters", Page 620

Tool change point G14

With the address **G14**, you program a movement of the slide to the saved tool change position at the end of the cycle.

Further information: "Defining the tool change position", Page 144

You can influence the approach to the tool change point as follows:

- No axis (do not approach the tool change position)
- 0: Simultaneously (default)
- 1: First X, then Z
- 2: First Z, then X
- 3: Only X
- 4: Only Z
- 5: Only Y (machine-dependent)
- 6: Simultaneous w/ Y (machine-dependent)

Cutting limits SX, SZ

With the addresses **SX** and **SZ** you limit the contour area to be machined in the X and Z directions. Seen from the tool position at the beginning of the cycle, the contour to be machined is truncated at these positions.

Additive compensation Dxx

With the address **Dxx** you activate an additive compensation for the entire cycle run. The xx stands for the compensation numbers 1 to 16. The additive compensation is switched off again at the end of the cycle.
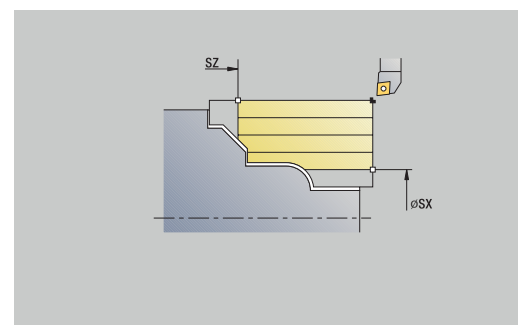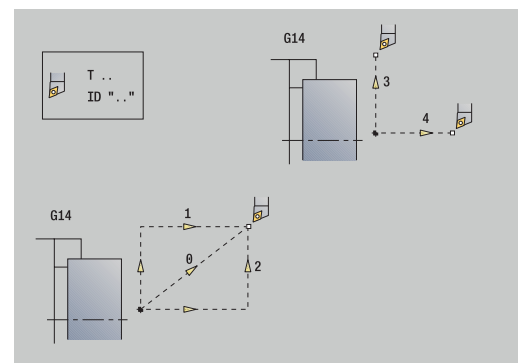
7.2 Workpiece blank cycles

Menu item	Meaning
	With the workpiece blank cycles you can describe the workpiece blank and the setup used. The workpiece blank cycles do not influence the machining process.
	The contours of workpiece blanks are shown during the simulation of the machining process.

Menu item	Workpiece blank cycles
	Bar/tube blank
	Defining a standard workpiece blank
	ICP workpiece blank contour
	Free workpiece blank description with ICP

Bar/tube blank

▶ Select **Define workpiece blank**

▶ Select **Bar/tube blank**

The cycle describes the workpiece blank and the setup (clamping) used. This information is evaluated in the **Simulation** submode.

Cycle parameters:

- **X**: **Outside diameter**
- **Z**: **Length** – Including transverse oversize and clamping range
- **I**: **Inside diameter**
- **K**: **Righthnd edge** – Transverse oversize
- **B**: **Clamp range**
- **J**: **Clamp type**
  - **0: Not clamped**
  - **1: Externally clamped**
  - **2: Internally clamped**
- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
  - Main drive
  - Opposing spindle for rear-face machining
- **RG**: **Activate contour follow-up** – Contour follow-up for **Teach-in** submode
  - **0: Without contour follow-up**
  - **1: With contour follow-up**

Further information: "Contour follow-up in Teach-in submode", Page 198
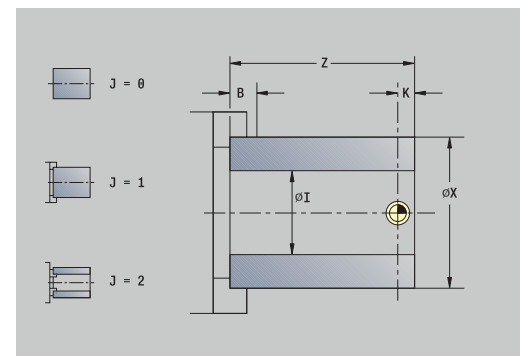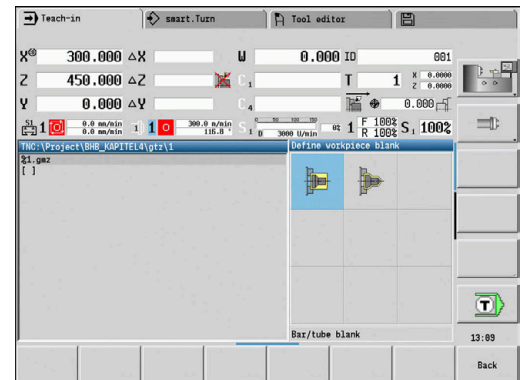
ICP workpiece blank contour

▶ Select **Define workpiece blank**

▶ Select **ICP workpiece blank contour**

The cycle describes the workpiece blank and the setup (clamping) used. This information is evaluated in the **Simulation** submode.

Cycle parameters:

- **X**: **Chuck diameter**
- **Z**: **Chuck position in Z**
- **B**: **Clamp range**
- **J**: **Clamp type**
  - **0: Not clamped**
  - **1: Externally clamped**
  - **2: Internally clamped**
- **RK**: **ICP contour number**
- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
  - Main drive
  - Opposing spindle for rear-face machining
- **RG**: **Activate contour follow-up** – Contour follow-up for **Teach-in** submode
  - **0: Without contour follow-up**
  - **1: With contour follow-up**

Further information: "Contour follow-up in Teach-in submode", Page 198
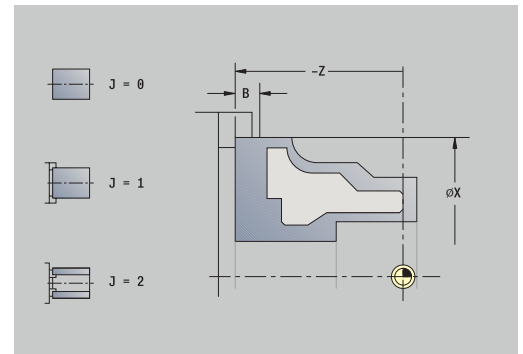
7.3 Single cut cycles

Menu item	Meaning
	In the single cut cycles you position the tool in rapid traverse, perform linear or circular cuts, machine chamfers or rounding arcs, and enter M functions.

Menu item	Single cut cycles
	Rapid positioning
T-Change approach	**Tool change point**
	Linear mach. longit./**Linear mach. transv.**: Single longitudinal/transverse cut
	Linear machining in angle: Single oblique cut
	Circular machining: Single circular cut (cutting direction depends on the selected menu item)
	Cham: Machine a chamfer
	Round.: Machine a rounding
	Call an **M function**
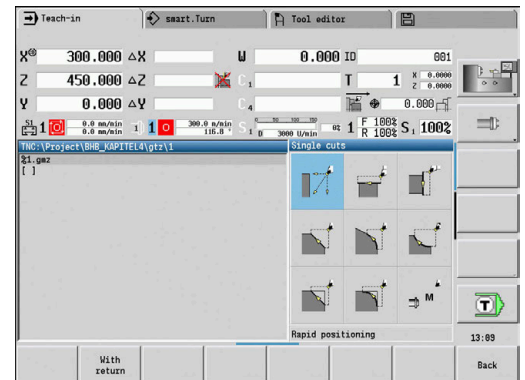
Rap. trav. positioning

▶ Select **Single cuts**

▶ Select **Rap. trav. positioning**

The tool moves at rapid traverse from the **Start point** to the **Target point**.

Cycle parameters:

- **X**, **Z**: **Start point**
- **X2**, **Z2**: **Target point**
- **T**: **Tool number** – turret pocket number
- **ID**: **ID no.**
- **MT**: **M after T**: **M** function that is executed after the **T** tool call
- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step
- **MFE**: **M at end**: **M** function that is executed at the end of the machining step
- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
  - Main drive
  - Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)
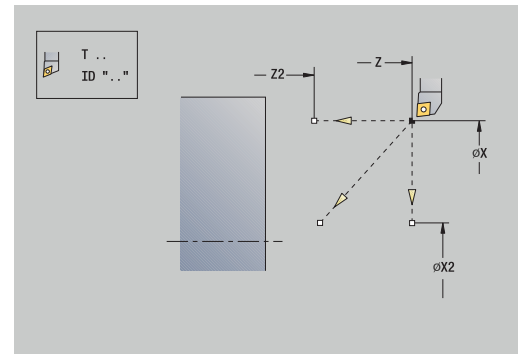
Approaching the tool change position

▶ Select **Single cuts**

▶ Select **Rap. trav. positioning**
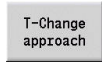
▶ Press the **T-Change approach** soft key

The tool moves at rapid traverse from the current position to the **Tool change point**.

Further information: "Tool change point G14", Page 203

After reaching the tool change point, the control switches to the tool indicated in **T**.

Cycle parameters:

- **G14**: **Tool change point** – Sequence (default: 0)
    - **0: Simultaneously** (diagonal path of traverse)
    - **1: First X, then Z**
    - **2: First Z, then X**
    - **3: Only X**
    - **4: Only Z**
    - **5: Only Y** (machine-dependent)
    - **6: Simultaneous w/ Y** (machine-dependent)
- **T**: **Tool number** – turret pocket number
- **ID**: **ID no.**
- **MT**: **M after T**: **M** function that is executed after the **T** tool call
- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step
- **MFE**: **M at end**: **M** function that is executed at the end of the machining step
- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
    - Main drive
    - Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)
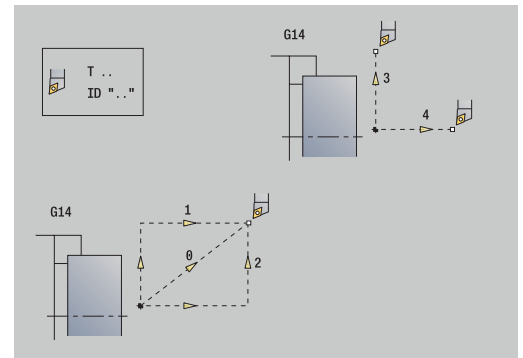
Linear mach. longit.

▶ Select **Single cuts**

▶ Select **Linear mach. longit.**

With return

▶ Select the scope of the cycle:
  - **Off**: When the cycle is completed, the tool remains at the cycle end position
  - **On**: The tool returns to the starting point

Linear mach. longit.: The tool moves from the **Start point** to the **End point contour Z2** at the programmed feed rate and remains at the cycle end position.

Linear mach. longit. (**With return**): The tool approaches the workpiece, executes the longitudinal cut and returns to the **Start point** at the end of cycle.

Cycle parameters:
- **X**, **Z**: **Start point**
- **X1**: **Start point contour** (if **With return** is active)
- **Z2**: **End point contour**
- **T**: **Tool number** – turret pocket number
- **G14**: **Tool change point** (if **With return** is active)
- **ID**: **ID no.**
- **S**: **Cutting speed** or **Constant speed**
- **F**: **Feed per revolution**
- **MT**: **M after T**: **M** function that is executed after the **T** tool call
- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step
- **MFE**: **M at end**: **M** function that is executed at the end of the machining step
- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
  - Main drive
  - Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)

> **i** Type of machining for technology database access: **Finishing**

Cycle run if **With return** is active:
1. Move from **Start point** to **Start point contour X1**
2. Move at programmed feed rate to **End point contour Z2**
3. Retract and return to **Start point** along paraxial path

Transv. linear maching

> ▶ Select **Single cuts**

> ▶ Select **Transv. linear maching**

With
return

> ▶ Select the scope of the cycle:
>   ■ **Off**: When the cycle is completed, the tool remains at the cycle end position
>   ■ **On**: The tool returns to the starting point

Transv. linear maching: The tool moves from the **Start point** to the **End point contour X2** at the programmed feed rate and remains at the cycle end position.

Transv. linear maching (**With return**): The tool approaches the workpiece, executes the transverse cut and returns to the **Start point** at the end of cycle.

Cycle parameters:
- **X**, **Z**: **Start point**
- **Z1**: **Start point contour** (if **With return** is active)
- **X2**: **End point contour**
- **T**: **Tool number** – turret pocket number
- **G14**: **Tool change point** (if **With return** is active)
- **ID**: **ID no.**
- **S**: **Cutting speed** or **Constant speed**
- **F**: **Feed per revolution**
- **MT**: **M after T**: **M** function that is executed after the **T** tool call
- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step
- **MFE**: **M at end**: **M** function that is executed at the end of the machining step
- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
   - Main drive
   - Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)

> **i** Type of machining for technology database access: **Finishing**

Cycle run if **With return** is active:
1 Move from **Start point** to **Start point contour Z1**
2 Move at programmed feed rate to **End point contour X2**
3 Retract and return to **Start point** along paraxial path

Linear machining in angle

▶ Select **Single cuts**

▶ Select **Linear machining in angle**

▶ Select the scope of the cycle:
- **Off**: When the cycle is completed, the tool remains at the cycle end position
- **On**: The tool returns to the starting point

Linear machining at angle: The control calculates the **target position** and moves the tool along a straight line from the **Start point** to the **Zielposition** at the programmed feed rate. When the cycle is completed, the tool remains at the cycle end position.

Linear machining at angle (**With return**): The control calculates the **Target position**. The tool then approaches the workpiece, executes the linear cut and returns to the **Start point** at the end of cycle. Tool-tip radius compensation is taken into account.

Cycle parameters:
- **X**, **Z**: **Start point**
- **X1**, **Z1**: **Start point contour** (if **With return** is active)
- **X2**, **Z2**: **End point contour**
- **A**: **Start angle** (range: −180° < **A** < 180°)
- **G47**: **Safety clearance** (if **With return** is active)
- **T**: **Tool number** – turret pocket number
- **G14**: **Tool change point** (if **With return** is active)
- **ID**: **ID no.**
- **S**: **Cutting speed** or **Constant speed**
- **F**: **Feed per revolution**
- **MT**: **M after T**: **M** function that is executed after the **T** tool call
- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step
- **MFE**: **M at end**: **M** function that is executed at the end of the machining step
- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
  - Main drive
  - Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)

> **ⓘ** Type of machining for technology database access:
> **Finishing**

> **ⓘ** Parameter combinations for defining the target point: see help graphic

Cycle run if **With return** is active:

1 Calculate **Target position**
2 Move along a linear path from **Start point** to **Start point contour X1**, **Z1**
3 Move to **Target position** at programmed feed rate
4 Retract and return to **Start point** along paraxial path

Circular machining

> ▶ Select **Single cuts**

> ▶ Select **Circular machining** (counterclockwise)

> ▶ Alternative: Select **Circular machining** (clockwise)
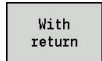
> ▶ Select the scope of the cycle:
> - **Off**: When the cycle is completed, the tool remains at the cycle end position
> - **On**: The tool returns to the starting point

Circular machining: The tool moves along a circular arc from the **Start point X**, **Z** to the **End point contour X2**, **Z2** at the programmed feed rate and remains at the cycle end position.

Circular machining (**With return**): The tool approaches the workpiece, executes the circular cut and returns to the **Start point** at the end of cycle. Tool-tip radius compensation is taken into account.

Cycle parameters:

- **X**, **Z**: **Start point**
- **X1**, **Z1**: **Start point contour** (if **With return** is active)
- **X2**, **Z2**: **End point contour**
- **R**: **Radius**
- **G47**: **Safety clearance** (if **With return** is active)
- **T**: **Tool number** – turret pocket number
- **G14**: **Tool change point** (if **With return** is active)
- **ID**: **ID no.**
- **S**: **Cutting speed** or **Constant speed**
- **F**: **Feed per revolution**
- **MT**: **M after T**: **M** function that is executed after the **T** tool call
- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step
- **MFE**: **M at end**: **M** function that is executed at the end of the machining step
- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
  - Main drive
  - Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)

> ℹ Type of machining for technology database access: **Finishing**

Cycle run if **With return** is active:

1 Move along a paraxial path from the **Start point** to the **Start point contour X1**, **Z1**

2 Move along a circular arc to the **End point contour X2**, **Z2**

3 Retract and return to **Start point** along paraxial path

Chamfer

▶ Select **Single cuts**

▶ Select **Chamfer**
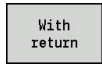
With return

▶ Select the scope of the cycle:
- **Off**: When the cycle is completed, the tool remains at the cycle end position
- **On**: The tool returns to the starting point

Chamfer: The cycle produces a chamfer that is dimensioned relative to the corner of the workpiece contour. When the cycle is completed, the tool remains at the cycle end position.

Chamfer (**With return**): The tool approaches the workpiece, machines the chamfer with dimensions relative to the corner of the workpiece contour and returns to the **Start point** at the end of cycle. Tool-tip radius compensation is taken into account.

Cycle parameters:
- **X**, **Z**: **Start point**
- **X1**, **Z1**: **Contour corner**
- **A**: **Start angle** – Angle of the chamfer (range: 0° < **A** < 90°)
- **I**, **K**: **Chamfer width** in X and Z
- **J**: **Position** (default: 1)
  The algebraic sign determines the cutting direction (see help graphics).
- **G47**: **Safety clearance** (if **With return** is active)
- **T**: **Tool number** – turret pocket number
- **G14**: **Tool change point** (if **With return** is active)
- **ID**: **ID no.**
- **S**: **Cutting speed** or **Constant speed**
- **F**: **Feed per revolution**
- **MT**: **M after T**: **M** function that is executed after the **T** tool call
- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step
- **MFE**: **M at end**: **M** function that is executed at the end of the machining step
- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
  - Main drive
  - Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)

> **i** Type of machining for technology database access:
> **Finishing**

Parameter combinations for defining the chamfer:

- **I** or **K** (45° chamfer)
- **I**, **K**
- **I**, **A** or **K**, **A**

Cycle run if **With return** is active:

1 Calculate chamfer **starting point** and chamfer **end point**
2 Move along a paraxial path from the **Start point** to the chamfer **starting point**
3 Move to chamfer **end point** at programmed feed rate
4 Retract and return to **Start point** along paraxial path

Rounding

▶ Select **Single cuts**

▶ Select **Rounding**
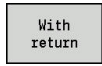
With
return

▶ Select the scope of the cycle:
  - **Off**: When the cycle is completed, the tool remains at the cycle end position
  - **On**: The tool returns to the starting point

Rounding: The cycle produces a rounding that is dimensioned relative to the corner of the workpiece contour. When the cycle is completed, the tool remains at the cycle end position.

Rounding (**With return**): The tool approaches the workpiece, machines the rounding with dimensions relative to the corner of the workpiece contour and returns to the **Start point** at the end of cycle. Tool-tip radius compensation is taken into account.

Cycle parameters:

- **X**, **Z**: **Start point**
- **X1**, **Z1**: **Contour corner**
- **R**: **Rounding**
- **J**: **Position** (default: 1)
  The algebraic sign determines the cutting direction (see help graphics).
- **G47**: **Safety clearance** (if **With return** is active)
- **T**: **Tool number** – turret pocket number
- **G14**: **Tool change point** (if **With return** is active)
- **ID**: **ID no.**
- **S**: **Cutting speed** or **Constant speed**
- **F**: **Feed per revolution**
- **MT**: **M after T**: **M** function that is executed after the **T** tool call
- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step
- **MFE**: **M at end**: **M** function that is executed at the end of the machining step
- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
  - Main drive
  - Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)

> 🛈 Type of machining for technology database access:
> **Finishing**

Cycle run if **With return** is active:

1 Calculate **starting point** and **end point** of rounding
2 Move along paraxial path from the **Start point** to the rounding **starting point**
3 Move along circular arc to rounding **end point** at programmed feed rate.
4 Retract and return to **Start point** along paraxial path

M functions

Machine commands (**M** functions) are only executed after **NC start** has been pressed. The **M list** soft key allows you to open an overview of the available **M** functions. For the meaning of the **M** functions, refer to your machine manual.

M function:

▶ Select **Single cuts**
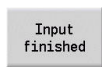
▶ Select **M function**

▶ Enter the number of the **M** function

▶ Confirm your input

▶ Press the **NC start** key

Spindle stop **M19** (spindle positioning):

▶ Select **Single cuts**

▶ Select **M function**

▶ Activate **M19**
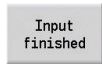
▶ Enter the stopping angle

▶ Confirm your input

▶ Press the **NC start** key

7.4 Turning cycles

Menu item	Meaning
	Turning cycles can be used for roughing and finishing of simple contours in **basic mode** and complex contours in **expanded mode**

Further information: "ICP contours", Page 436

> ℹ️
> - **Proportioning of cuts**: The control calculates an infeed that is <= **Feed depth P**. **Abrasive cuts** are thus avoided
> - **Oversizes** are considered in **expanded mode**
> - **Tool-tip radius compensation** is active
> - **Safety clearance** after each cut:
>   - Basic mode: 1 mm
>   - Expanded mode: The safety clearance is set separately for inside and outside machining
>   **Further information:** "List of machine parameters", Page 620

Cutting and infeed directions for turning cycles: The control determines the cutting and infeed directions from the cycle parameters.

The decisive ones are:

- **Basic mode**: The parameters for **Starting point X**, **Z** (in **Machine** operating mode: current tool position) and **Start point contour X1** / **End point contour Z2**
- **Expanded mode**: The parameters for **Start point contour X1**, **Z1** and **End point contour X2**, **Z2**
- **ICP cycles**: The parameters for **Starting point X**, **Z** (in **Machine** operating mode: current tool position) and starting point of ICP contour

Menu item		Turning cycles
		Cut longitud./Cut transv.
		Roughing and finishing cycle for simple contours
		Plunge longit./Plunge transv.
		Roughing and finishing cycle for simple plunge-cut contours
		ICP-long.parallel contour/ICP-transv.parallel contour
		Roughing and finishing cycle for any type of contour (cutting paths parallel to finished part)
		ICP cut longitud./ICP cut transv.
		Roughing and finishing cycle for any type of contour
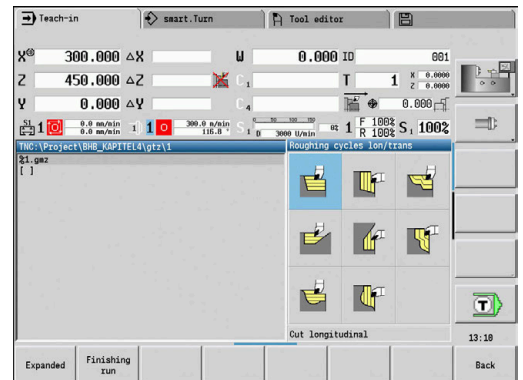
Tool position

It is important that you observe the **Start point X**, **Z** tool position before executing any of the turning cycles in expanded mode.

The rules apply for all cutting and infeed directions as well as for roughing and finishing:

- The starting point must not be located in the shaded area
- The area to be machined starts at the **Start point X**, **Z** if the tool is positioned **before** the contour area. The control will otherwise only machine the contour area defined
- If the **Start point X**, **Z** for inside machining is located above the turning center, only the contour area defined will be machined

(**A** = **Start point contour X1**, **Z1**; **E** = **End point contour X2**, **Z2**)

Contour elements

Contour elements in turning cycles

	Basic mode Machining a rectangular area
	Expanded mode Oblique cut at contour start
	Expanded mode Oblique cut at contour end
	Expanded mode Oblique cuts at contour start and end with angles > 45°
	Expanded mode **One** oblique cut (by entering the starting point of contour, end point of contour and starting angle)
	Expanded mode Rounding arc
	Expanded mode Chamfer (or rounding) at contour end
	Basic mode Machining with descending contour
	Basic mode Oblique cut at contour end
	Expanded mode Rounding in contour valley (in both corners)
	Expanded mode Chamfer (or rounding) at contour start
	Expanded mode Chamfer (or rounding) at contour end
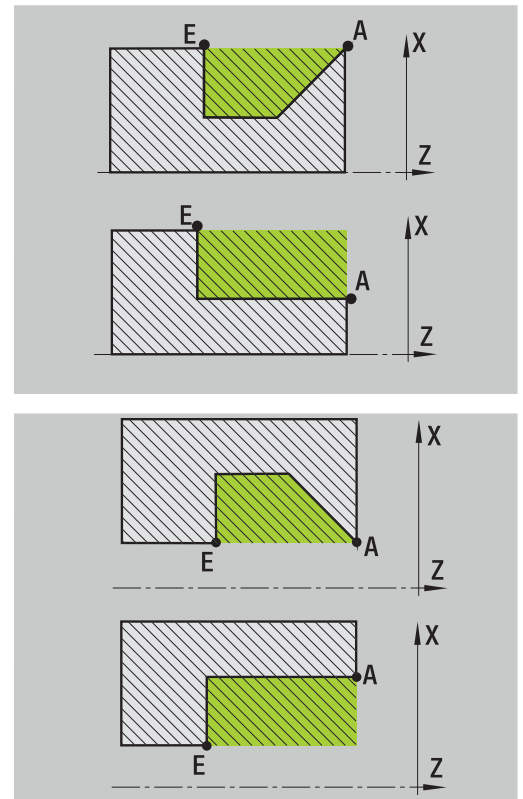
Cut longitud.

▶ Select **Roughing cycles lon/trans**

▶ Select **Cut longitud.**

The cycle roughs the rectangle described by the **Start point** and the **Start point contour X1**/**End point contour Z2**.

Cycle parameters:

- **X**, **Z**: **Start point**
- **X1**: **Start point contour**
- **Z2**: **End point contour**
- **P**: **Feed depth** – Maximum infeed depth
- **H**: **Contour smoothing**
  - **0: With each cut**
  - **1: With the last cut**
  - **2: No smoothing**
- **G47**: **Safety clearance**
  Further information: "Safety clearance G47", Page 203
- **T**: **Tool number** – turret pocket number
- **G14**: **Tool change point**
  Further information: "Tool change point G14", Page 203
- **ID**: **ID no.**
- **S**: **Cutting speed** or **Constant speed**
- **F**: **Feed per revolution**
- **MT**: **M after T**: **M** function that is executed after the **T** tool call
- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step
- **MFE**: **M at end**: **M** function that is executed at the end of the machining step
- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
  - Main drive
  - Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)

> ℹ️ Type of machining for technology database access:
> **Roughing**

Cycle run:

1 Calculate the proportioning of cuts (infeed)
2 Approach the workpiece from the **Start point** for the first pass
3 Move to the **End point contour Z2** at the programmed feed rate
4 The contour is machined according to the setting in **Contour smoothing H**
5 Retract and approach for the next pass
6 Repeat 3 to 5 until the **Start point contour X1** is reached
7 Return to the **Start point** on a diagonal path
8 Move to the **Tool change point** according to the setting in **G14**

Cut transv.

▶ Select **Roughing cycles lon/trans**

▶ Select **Cut transv.**

The cycle roughs the rectangle described by the **Start point** and the **Start point contour Z1**/**End point contour X2**.

Cycle parameters:
- **X**, **Z**: **Start point**
- **Z1**: **Start point contour**
- **X2**: **End point contour**
- **P**: **Feed depth** – Maximum infeed depth
- **H**: **Contour smoothing**
    - **0: With each cut**
    - **1: With the last cut**
    - **2: No smoothing**
- **G47**: **Safety clearance**
  Further information: "Safety clearance G47", Page 203
- **T**: **Tool number** – turret pocket number
- **G14**: **Tool change point**
  Further information: "Tool change point G14", Page 203
- **ID**: **ID no.**
- **S**: **Cutting speed** or **Constant speed**
- **F**: **Feed per revolution**
- **MT**: **M after T**: **M** function that is executed after the **T** tool call
- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step
- **MFE**: **M at end**: **M** function that is executed at the end of the machining step
- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
    - Main drive
    - Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)

> **i** Type of machining for technology database access:
> **Roughing**

Cycle run:

1 Calculate the proportioning of cuts (infeed)
2 Approach the workpiece from the **Start point** for the first pass
3 Move to the **End point contour X2** at the programmed feed rate
4 The contour is machined according to the setting in **Contour smoothing H**
5 Retract and approach for the next pass
6 Repeat 3 to 5 until the **Start point contour Z1** is reached
7 Return to the **Start point** on a diagonal path
8 Move to the **Tool change point** according to the setting in **G14**

Cut longitud. − Expanded

▶ Select **Roughing cycles lon/trans**

▶ Select **Cut longitud.**

Expanded

▶ Select the **Expanded** soft key

Taking the oversizes into account, the cycle roughs the area described by the **Start point** and the **Start point contour X1**/**End point contour Z2**.

Cycle parameters:

- **X**, **Z**: **Start point**
- **X1**, **Z1**: **Start point contour**
- **X2**, **Z2**: **End point contour**
- **P**: **Feed depth** − Maximum infeed depth
- **H**: **Contour smoothing**
    - **0: With each cut**
    - **1: With the last cut**
    - **2: No smoothing**
- **I**, **K**: **O-size X** and **Z**
- **A**: **Start angle** (range: 0° <= **A** < 90°; default: 0°)
- **W**: **Final angle** (range: 0° <= **W** < 90°; default: 0°)
- **R**: **Rounding**
- **T**: **Tool number** − turret pocket number
- **G14**: **Tool change point**
  Further information: "Tool change point G14", Page 203
- **ID**: **ID no.**
- **S**: **Cutting speed** or **Constant speed**
- **F**: **Feed per revolution**
- **B1**, **B2**: **-B cham./+B round.** (**B1** at contour start and **B2** at contour end)
    - **B** > 0: Radius of rounding
    - **B** < 0: Width of chamfer
- **BP**: **Interval time** − time during which the feed motion will be interrupted
  The chip is broken by the intermittent feed.
- **BF**: **Feed period** − time interval until the next break
  The chip is broken by the intermittent feed.
- **G47**: **Safety clearance**
  Further information: "Safety clearance G47", Page 203
- **MT**: **M after T**: **M** function that is executed after the **T** tool call
- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step
- **MFE**: **M at end**: **M** function that is executed at the end of the machining step
- **WP**: **No. of spindle** − Displays which workpiece spindle is used to execute the cycle (machine-dependent)

- Main drive
- Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)

> **i** Type of machining for technology database access:
> **Roughing**

Cycle run:

1. Calculate the proportioning of cuts (infeed)
2. Approach the workpiece from the **Start point** for the first pass
3. Move at the programmed feed rate to the **End point contour Z2** or, if defined, to one of the optional contour elements
4. The contour is machined according to the setting in **Contour smoothing H**
5. Retract and approach for the next pass
6. Repeat 3 to 5 until the **Start point contour X1** is reached
7. Return to the **Start point** on a paraxial path
8. Move to the **Tool change point** according to the setting in **G14**

Cut transv. – Expanded

▶ Select **Roughing cycles lon/trans**

▶ Select **Cut transv.**

▶ Select the **Expanded** soft key

Taking the oversizes into account, the cycle roughs the area described by the **Start point** and the **Start point contour Z1**/**End point contour X2**.

Cycle parameters:

- **X**, **Z**: **Start point**
- **X1**, **Z1**: **Start point contour**
- **X2**, **Z2**: **End point contour**
- **P**: **Feed depth** – Maximum infeed depth
- **H**: **Contour smoothing**
    - **0: With each cut**
    - **1: With the last cut**
    - **2: No smoothing**
- **I**, **K**: **O-size X** and **Z**
- **A**: **Start angle** (range: 0° <= **A** < 90°; default: 0°)
- **W**: **Final angle** (range: 0° <= **W** < 90°; default: 0°)
- **R**: **Rounding**
- **T**: **Tool number** – turret pocket number
- **G14**: **Tool change point**
  Further information: "Tool change point G14", Page 203
- **ID**: **ID no.**
- **S**: **Cutting speed** or **Constant speed**
- **F**: **Feed per revolution**
- **B1**, **B2**: **-B cham./+B round.** (**B1** at contour start and **B2** at contour end)
    - **B** > 0: Radius of rounding
    - **B** < 0: Width of chamfer
- **BP**: **Interval time** – time during which the feed motion will be interrupted
  The chip is broken by the intermittent feed.
- **BF**: **Feed period** – time interval until the next break
  The chip is broken by the intermittent feed.
- **G47**: **Safety clearance**
  Further information: "Safety clearance G47", Page 203
- **MT**: **M after T**: **M** function that is executed after the **T** tool call
- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step
- **MFE**: **M at end**: **M** function that is executed at the end of the machining step
- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)

- Main drive
- Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)

> **i** Type of machining for technology database access:
> **Roughing**

Cycle run:
1 Calculate the proportioning of cuts (infeed)
2 Approach the workpiece from the **Start point** for the first pass
3 Move at the programmed feed rate to the **End point contour X2** or, if defined, to one of the optional contour elements
4 The contour is machined according to the setting in **Contour smoothing H**
5 Retract and approach for the next pass
6 Repeat 3 to 5 until the **Start point contour Z1** is reached
7 Return to the **Start point** on a paraxial path
8 Move to the **Tool change point** according to the setting in **G14**

Finishing cut longitud.

▶ Select **Roughing cycles lon/trans**

▶ Select **Cut longitud.**

▶ Press the **Finishing run** soft key

The cycle finishes the contour area from **Start point contour X1** to **End point contour Z2**.

> 🛈 At the end of the cycle, the tool returns to the **Start point**.

Cycle parameters:

- **X**, **Z**: **Start point**
- **X1**: **Start point contour**
- **Z2**: **End point contour**
- **G47**: **Safety clearance**
  Further information: "Safety clearance G47", Page 203
- **T**: **Tool number** – turret pocket number
- **G14**: **Tool change point**
  Further information: "Tool change point G14", Page 203
- **ID**: **ID no.**
- **S**: **Cutting speed** or **Constant speed**
- **F**: **Feed per revolution**
- **MT**: **M after T**: **M** function that is executed after the **T** tool call
- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step
- **MFE**: **M at end**: **M** function that is executed at the end of the machining step
- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
  - Main drive
  - Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)

> 🛈 Type of machining for technology database access:
> **Finishing**

Cycle run:

1 Move in transverse direction from the **Start point** to the **Start point contour X1**
2 Finish first in longitudinal direction, then in transverse direction
3 Return in longitudinal direction to the **Start point**
4 Move to the **Tool change point** according to the setting in **G14**

Finishing cut transv.

> ▶ Select **Roughing cycles lon/trans**

> ▶ Select **Cut transv.**

Finishing run

> ▶ Press the **Finishing run** soft key

The cycle finishes the contour area from **Start point contour Z1** to **End point contour X2**.

> **i** At the end of the cycle, the tool returns to the **Start point**.

Cycle parameters:

- **X**, **Z**: **Start point**
- **Z1**: **Start point contour**
- **X2**: **End point contour**
- **G47**: **Safety clearance**
  Further information: "Safety clearance G47", Page 203
- **T**: **Tool number** – turret pocket number
- **G14**: **Tool change point**
  Further information: "Tool change point G14", Page 203
- **ID**: **ID no.**
- **S**: **Cutting speed** or **Constant speed**
- **F**: **Feed per revolution**
- **MT**: **M after T**: **M** function that is executed after the **T** tool call
- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step
- **MFE**: **M at end**: **M** function that is executed at the end of the machining step
- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
  - Main drive
  - Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)

> **i** Type of machining for technology database access:
> **Finishing**

Cycle run:

1 Move in longitudinal direction from **Start point** to **Start point contour Z1**
2 Finish first in transverse direction, then in longitudinal direction
3 Return in transverse direction to the **Start point**
4 Move to the **Tool change point** according to the setting in **G14**

Finishing cut longitud. – Expanded

▶ Select **Roughing cycles lon/trans**

▶ Select **Cut longitud.**

Expanded

▶ Select the **Expanded** soft key

Finishing run

▶ Press the **Finishing run** soft key

The cycle finishes the contour area from **Start point contour** to **End point contour**.

> ℹ️ When the cycle is completed, the tool remains at the cycle end position.

Cycle parameters:
- **X**, **Z**: **Start point**
- **X1**, **Z1**: **Start point contour**
- **X2**, **Z2**: **End point contour**
- **DXX**: **Add. correction number** (range: 1 to 16) **Further information:** "Additive compensation Dxx", Page 203
- **G58**: **Contour-parallel oversize**
- **A**: **Start angle** (range: 0° <= **A** < 90°; default: 0°)
- **W**: **Final angle** (range: 0° <= **W** < 90°; default: 0°)
- **R**: **Rounding**
- **T**: **Tool number** – turret pocket number
- **G14**: **Tool change point**
  Further information: "Tool change point G14", Page 203
- **ID**: **ID no.**
- **S**: **Cutting speed** or **Constant speed**
- **F**: **Feed per revolution**
- **B1**, **B2**: **-B cham./+B round.** (**B1** at contour start and **B2** at contour end)
  - **B** > 0: Radius of rounding
  - **B** < 0: Width of chamfer
- **G47**: **Safety clearance**
  Further information: "Safety clearance G47", Page 203
- **MT**: **M after T**: **M** function that is executed after the **T** tool call
- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step
- **MFE**: **M at end**: **M** function that is executed at the end of the machining step
- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
  - Main drive
  - Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)

■ **HC**: **Shoe brake** (machine-dependent)

■ **DF**: **Miscellaneous function** (machine-dependent)

> ⓘ Type of machining for technology database access:
> **Finishing**

Cycle run:

1   Move in transverse direction from the **Start point** to the **Start point contour X1**, **Z1**

2   Finish the contour area from the **Start point contour X1**, **Z1** to the **End point contour X2**, **Z2**, taking the optional contour elements into account

3   Move to the **Tool change point** according to the setting in **G14**

Finishing cut transv. – Expanded

▶ Select **Roughing cycles lon/trans**

▶ Select **Cut transv.**

▶ Select the **Expanded** soft key

▶ Press the **Finishing run** soft key

The cycle finishes the contour area from **Start point contour** to **End point contour**.

i When the cycle is completed, the tool remains at the cycle end position.

Cycle parameters:
- **X**, **Z**: **Start point**
- **X1**, **Z1**: **Start point contour**
- **X2**, **Z2**: **End point contour**
- **DXX**: **Add. correction number** (range: 1 to 16) **Further information:** "Additive compensation Dxx", Page 203
- **G58**: **Contour-parallel oversize**
- **A**: **Start angle** (range: 0° <= **A** < 90°; default: 0°)
- **W**: **Final angle** (range: 0° <= **W** < 90°; default: 0°)
- **R**: **Rounding**
- **T**: **Tool number** – turret pocket number
- **G14**: **Tool change point**
  Further information: "Tool change point G14", Page 203
- **ID**: **ID no.**
- **S**: **Cutting speed** or **Constant speed**
- **F**: **Feed per revolution**
- **B1**, **B2**: **-B cham./+B round.** (**B1** at contour start and **B2** at contour end)
  - **B** > 0: Radius of rounding
  - **B** < 0: Width of chamfer
- **G47**: **Safety clearance**
  Further information: "Safety clearance G47", Page 203
- **MT**: **M after T**: **M** function that is executed after the **T** tool call
- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step
- **MFE**: **M at end**: **M** function that is executed at the end of the machining step
- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
  - Main drive
  - Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)

- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)

> ℹ️ Type of machining for technology database access:
> **Finishing**

Cycle run:

1. Move in longitudinal direction from the **Start point** to the **Start point contour X1**, **Z1**
2. Finish the contour area from the **Start point contour X1**, **Z1** to the **End point contour X2**, **Z2**, taking the optional contour elements into account
3. Move to the **Tool change point** according to the setting in **G14**

Teach-in | Turning cycles

Cutting, longitudinal plunge

▶ Select **Roughing cycles lon/trans**

▶ Select **Plunge longit.**

The cycle roughs the area described by the **Start point contour**, **End point contour** and **Plunging angle**.

> ℹ️ ■ The tool plunges with the maximum possible angle, leaving material remaining
> ■ The steeper the tool plunges into the material, the greater the feed rate decrease (max. 50 %)

Cycle parameters:
- **X**, **Z**: **Start point**
- **X1**, **Z1**: **Start point contour**
- **X2**, **Z2**: **End point contour**
- **P**: **Feed depth** – Maximum infeed depth
- **H**: **Contour smoothing**
    - **0: With each cut**
    - **1: With the last cut**
    - **2: No smoothing**
- **A**: **Plunging angle** (range: 0° <= **A** < 90°; default: 0°)
- **W**: **Final angle** (range: 0° <= **W** < 90°; default: 0°)
- **G47**: **Safety clearance**
  Further information: "Safety clearance G47", Page 203
- **T**: **Tool number** – turret pocket number
- **G14**: **Tool change point**
  Further information: "Tool change point G14", Page 203
- **ID**: **ID no.**
- **S**: **Cutting speed** or **Constant speed**
- **F**: **Feed per revolution**
- **MT**: **M after T**: **M** function that is executed after the **T** tool call
- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step
- **MFE**: **M at end**: **M** function that is executed at the end of the machining step
- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
    - Main drive
    - Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)

> ℹ️ Type of machining for technology database access:
> **Roughing**

Cycle run:

1　Calculate the proportioning of cuts (infeed)
2　Approach the workpiece from the **Start point** on a paraxial path for the first pass
3　Plunge-cut at **Plunging angle A** with reduced feed
4　Move at the programmed feed rate to the **End point contour Z2** or to the oblique surface defined by the **Final angle W**
5　The contour is machined according to the setting in **Contour smoothing H**
6　Return and approach again for the next pass
7　Repeat 3 to 6 until the **End point contour X2** is reached
8　Return to the **Start point** on a paraxial path
9　Move to the **Tool change point** according to the setting in **G14**

Cutting, transverse plunge

▶ Select **Roughing cycles lon/trans**

▶ Select **Plunge transv.**

The cycle roughs the area described by the **Start point contour**, **End point contour** and **Plunging angle**.

> ℹ️
> - The tool plunges with the maximum possible angle, leaving material remaining
> - The steeper the tool plunges into the material, the greater the feed rate decrease (max. 50 %)

Cycle parameters:
- **X**, **Z**: **Start point**
- **X1**, **Z1**: **Start point contour**
- **X2**, **Z2**: **End point contour**
- **P**: **Feed depth** – Maximum infeed depth
- **H**: **Contour smoothing**
    - **0: With each cut**
    - **1: With the last cut**
    - **2: No smoothing**
- **A**: **Plunging angle** (range: 0° <= **A** < 90°; default: 0°)
- **W**: **Final angle** (range: 0° <= **W** < 90°; default: 0°)
- **G47**: **Safety clearance**
  Further information: "Safety clearance G47", Page 203
- **T**: **Tool number** – turret pocket number
- **G14**: **Tool change point**
  Further information: "Tool change point G14", Page 203
- **ID**: **ID no.**
- **S**: **Cutting speed** or **Constant speed**
- **F**: **Feed per revolution**
- **MT**: **M after T**: **M** function that is executed after the **T** tool call
- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step
- **MFE**: **M at end**: **M** function that is executed at the end of the machining step
- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
    - Main drive
    - Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)

> ℹ️
> Type of machining for technology database access:
> **Roughing**

Cycle run:

1 Calculate the proportioning of cuts (infeed)
2 Approach the workpiece from the **Start point** on a paraxial path for the first pass
3 Plunge-cut at **Plunging angle A** with reduced feed
4 Move at the programmed feed rate to the **End point contour X2** or to the oblique surface defined by the **Final angle W**
5 The contour is machined according to the setting in **Contour smoothing H**
6 Return and approach again for the next pass
7 Repeat 3 to 6 until the **End point contour Z2** is reached
8 Return to the **Start point** on a paraxial path
9 Move to the **Tool change point** according to the setting in **G14**

Cutting, longitudinal plunge – Expanded

▶ Select **Roughing cycles lon/trans**

▶ Select **Plunge longit.**

Expanded

▶ Select the **Expanded** soft key

The cycle roughs the area described by the **Start point contour**, **End point contour** and **Plunging angle**, taking the oversizes into account.

> ℹ
> ■ The tool plunges with the maximum possible angle, leaving material remaining
> ■ The steeper the tool plunges into the material, the greater the feed rate decrease (max. 50 %)

Cycle parameters:
- **X**, **Z**: **Start point**
- **X1**, **Z1**: **Start point contour**
- **X2**, **Z2**: **End point contour**
- **P**: **Feed depth** – Maximum infeed depth
- **H**: **Contour smoothing**
    - **0: With each cut**
    - **1: With the last cut**
    - **2: No smoothing**
- **I**, **K**: **O-size X** and **Z**
- **A**: **Plunging angle** (range: 0° <= **A** < 90°; default: 0°)
- **W**: **Final angle** (range: 0° <= **W** < 90°; default: 0°)
- **R**: **Rounding**
- **T**: **Tool number** – turret pocket number
- **G14**: **Tool change point**
  Further information: "Tool change point G14", Page 203
- **ID**: **ID no.**
- **S**: **Cutting speed** or **Constant speed**
- **F**: **Feed per revolution**
- **B1**, **B2**: **-B cham./+B round.** (**B1** at contour start and **B2** at contour end)
- **BP**: **Interval time** – time during which the feed motion will be interrupted
  The chip is broken by the intermittent feed.
- **BF**: **Feed period** – time interval until the next break
  The chip is broken by the intermittent feed.
- **G47**: **Safety clearance**
  Further information: "Safety clearance G47", Page 203
- **MT**: **M after T**: **M** function that is executed after the **T** tool call
- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step
- **MFE**: **M at end**: **M** function that is executed at the end of the machining step

- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
  - Main drive
  - Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)

> **ⓘ** Type of machining for technology database access:
> **Roughing**

Cycle run:

1 Calculate the proportioning of cuts (infeed)
2 Approach the workpiece from the **Start point** along a paraxial path for the first pass
3 Plunge-cut at **Plunging angle A** at reduced feed rate
4 Move at the programmed feed rate to the **End point contour Z2** or, if defined, to one of the optional contour elements
5 The contour is machined according to the setting in **Contour smoothing H**
6 Retract and approach for the next pass
7 Repeat 3 to 6 until the **End point contour X2** is reached
8 Return to the **Start point** along a paraxial path
9 Move to the **Tool change point** according to the setting in **G14**

Cut transverse, plunge – Expanded

▶ Select **Roughing cycles lon/trans**

▶ Select **Plunge transv.**

Expanded

▶ Select the **Expanded** soft key

The cycle roughs the area described by the **Start point contour**, **End point contour** and **Plunging angle**, taking the oversizes into account.

> ℹ ▪ The tool plunges with the maximum possible angle, leaving material remaining
> ▪ The steeper the tool plunges into the material, the greater the feed rate decrease (max. 50 %)

Cycle parameters:

- **X**, **Z**: **Start point**
- **X1**, **Z1**: **Start point contour**
- **X2**, **Z2**: **End point contour**
- **P**: **Feed depth** – Maximum infeed depth
- **H**: **Contour smoothing**
    - **0: With each cut**
    - **1: With the last cut**
    - **2: No smoothing**
- **I**, **K**: **O-size X** and **Z**
- **A**: **Plunging angle** (range: 0° <= **A** < 90°; default: 0°)
- **W**: **Final angle** (range: 0° <= **W** < 90°; default: 0°)
- **R**: **Rounding**
- **T**: **Tool number** – turret pocket number
- **G14**: **Tool change point**
  Further information: "Tool change point G14", Page 203
- **ID**: **ID no.**
- **S**: **Cutting speed** or **Constant speed**
- **F**: **Feed per revolution**
- **B1**, **B2**: **-B cham./+B round.** (**B1** at contour start and **B2** at contour end)
- **BP**: **Interval time** – time during which the feed motion will be interrupted
  The chip is broken by the intermittent feed.
- **BF**: **Feed period** – time interval until the next break
  The chip is broken by the intermittent feed.
- **G47**: **Safety clearance**
  Further information: "Safety clearance G47", Page 203
- **MT**: **M after T**: **M** function that is executed after the **T** tool call
- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step
- **MFE**: **M at end**: **M** function that is executed at the end of the machining step

- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
  - Main drive
  - Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)

> **ⓘ** Type of machining for technology database access:
> **Roughing**

Cycle run:

1. Calculate the proportioning of cuts (infeed)
2. Approach the workpiece from the **Start point** along a paraxial path for the first pass
3. Plunge-cut at **Plunging angle A** at reduced feed rate
4. Move at the programmed feed rate to the **End point contour X2** or, if defined, to one of the optional contour elements
5. The contour is machined according to the setting in **Contour smoothing H**
6. Retract and approach for the next pass
7. Repeat 3 to 6 until the **End point contour Z2** is reached
8. Return to the **Start point** along a paraxial path
9. Move to the **Tool change point** according to the setting in **G14**

Longitudinal cutting, finishing plunge

▶ Select **Roughing cycles lon/trans**

▶ Select **Plunge longit.**

▶ Press the **Finishing run** soft key

The cycle finishes the contour area from **Start point contour** to **End point contour**. At the end of the cycle, the tool returns to the **Start point**.

> ● ■ The tool plunges with the maximum possible angle, leaving material remaining
> ■ The steeper the tool plunges into the material, the greater the feed rate decrease (max. 50 %)

Cycle parameters:
- **X**, **Z**: **Start point**
- **X1**, **Z1**: **Start point contour**
- **X2**, **Z2**: **End point contour**
- **A**: **Plunging angle** (range: 0° <= **A** < 90°; default: 0°)
- **W**: **Final angle** (range: 0° <= **W** < 90°; default: 0°)
- **G47**: **Safety clearance**
  Further information: "Safety clearance G47", Page 203
- **T**: **Tool number** – turret pocket number
- **G14**: **Tool change point**
  Further information: "Tool change point G14", Page 203
- **ID**: **ID no.**
- **S**: **Cutting speed** or **Constant speed**
- **F**: **Feed per revolution**
- **MT**: **M after T**: **M** function that is executed after the **T** tool call
- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step
- **MFE**: **M at end**: **M** function that is executed at the end of the machining step
- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
  - Main drive
  - Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)

> ● Type of machining for technology database access:
> **Finishing**

Cycle run:

1 Move in transverse direction from the **Start point** to the **Start point contour X1**, **Z1**
2 Finish the defined contour area
3 Return to the **Start point** along a paraxial path
4 Move to the **Tool change point** according to the setting in **G14**

Traverse cutting, finishing plunge

▶ Select **Roughing cycles lon/trans**

▶ Select **Plunge transv.**

▶ Press the **Finishing run** soft key

The cycle finishes the contour area from **Start point contour** to **End point contour**. At the end of the cycle, the tool returns to the **Start point**.

ℹ
- The tool plunges with the maximum possible angle, leaving material remaining
- The steeper the tool plunges into the material, the greater the feed rate decrease (max. 50 %)

Cycle parameters:
- **X**, **Z**: **Start point**
- **X1**, **Z1**: **Start point contour**
- **X2**, **Z2**: **End point contour**
- **A**: **Plunging angle** (range: 0° <= **A** < 90°; default: 0°)
- **W**: **Final angle** (range: 0° <= **W** < 90°; default: 0°)
- **G47**: **Safety clearance**
  Further information: "Safety clearance G47", Page 203
- **T**: **Tool number** – turret pocket number
- **G14**: **Tool change point**
  Further information: "Tool change point G14", Page 203
- **ID**: **ID no.**
- **S**: **Cutting speed** or **Constant speed**
- **F**: **Feed per revolution**
- **MT**: **M after T**: **M** function that is executed after the **T** tool call
- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step
- **MFE**: **M at end**: **M** function that is executed at the end of the machining step
- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
  - Main drive
  - Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)

ℹ Type of machining for technology database access:
Finishing

Cycle run:

1 Move in transverse direction from the **Start point** to the **Start point contour X1**, **Z1**
2 Finish the defined contour area
3 Return to the **Start point** along a paraxial path
4 Move to the **Tool change point** according to the setting in **G14**

Longitudinal cutting, finishing plunge – Expanded

▶ Select **Roughing cycles lon/trans**

▶ Select **Plunge longit.**

Expanded   ▶ Select the **Expanded** soft key

Finishing run   ▶ Press the **Finishing run** soft key

The cycle finishes the contour area from **Start point contour** to **End point contour**. When the cycle is completed, the tool remains at the cycle end position.

> ⓘ ▪ The tool plunges with the maximum possible angle, leaving material remaining
> ▪ The steeper the tool plunges into the material, the greater the feed rate decrease (max. 50 %)

Cycle parameters:
- **X**, **Z**: **Start point**
- **X1**, **Z1**: **Start point contour**
- **X2**, **Z2**: **End point contour**
- **DXX**: **Add. correction number** (range: 1 to 16) **Further information:** "Additive compensation Dxx", Page 203
- **G58**: **Contour-parallel oversize**
- **A**: **Plunging angle** (range: 0° <= **A** < 90°; default: 0°)
- **W**: **Final angle** (range: 0° <= **W** < 90°; default: 0°)
- **R**: **Rounding**
- **T**: **Tool number** – turret pocket number
- **G14**: **Tool change point**
  Further information: "Tool change point G14", Page 203
- **ID**: **ID no.**
- **S**: **Cutting speed** or **Constant speed**
- **F**: **Feed per revolution**
- **B1**, **B2**: **-B cham./+B round.** (**B1** at contour start and **B2** at contour end)
  - **B** > 0: Radius of rounding
  - **B** < 0: Width of chamfer
- **G47**: **Safety clearance**
  Further information: "Safety clearance G47", Page 203
- **MT**: **M after T**: **M** function that is executed after the **T** tool call
- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step

- **MFE**: **M at end**: **M** function that is executed at the end of the machining step
- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
  - Main drive
  - Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)

> **i** Type of machining for technology database access:
> **Finishing**

Cycle run:
1 Move along a paraxial path from the **Start point** to the **Start point contour X1**, **Z1**
2 Finish the defined contour area, taking the optional contour elements into account
3 Move to the **Tool change point** according to the setting in **G14**

Traverse cutting, finishing plunge – Expanded

▶ Select **Roughing cycles lon/trans**

▶ Select **Plunge transv.**

| Expanded |

▶ Select the **Expanded** soft key

| Finishing run |

▶ Press the **Finishing run** soft key

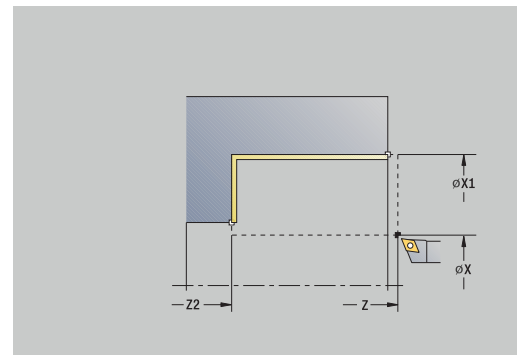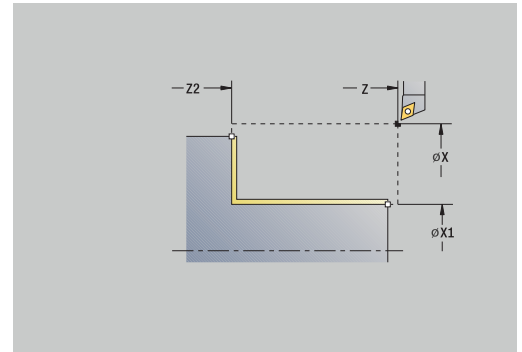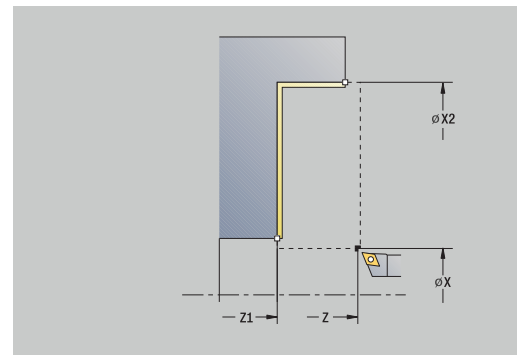The cycle finishes the contour area from **Start point contour** to **End point contour**. When the cycle is completed, the tool remains at the cycle end position.

> ⓘ ■ The tool plunges with the maximum possible angle, leaving material remaining
> ■ The steeper the tool plunges into the material, the greater the feed rate decrease (max. 50 %)

Cycle parameters:
- **X**, **Z**: **Start point**
- **X1**, **Z1**: **Start point contour**
- **X2**, **Z2**: **End point contour**
- **DXX**: **Add. correction number** (range: 1 to 16) **Further information:** "Additive compensation Dxx", Page 203
- **G58**: **Contour-parallel oversize**
- **A**: **Plunging angle** (range: 0° <= **A** < 90°; default: 0°)
- **W**: **Final angle** (range: 0° <= **W** < 90°; default: 0°)
- **R**: **Rounding**
- **T**: **Tool number** – turret pocket number
- **G14**: **Tool change point**
  Further information: "Tool change point G14", Page 203
- **ID**: **ID no.**
- **S**: **Cutting speed** or **Constant speed**
- **F**: **Feed per revolution**
- **B1**, **B2**: **-B cham./+B round.** (**B1** at contour start and **B2** at contour end)
  - **B** > 0: Radius of rounding
  - **B** < 0: Width of chamfer
- **G47**: **Safety clearance**
  Further information: "Safety clearance G47", Page 203
- **MT**: **M after T**: **M** function that is executed after the **T** tool call

- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step
- **MFE**: **M at end**: **M** function that is executed at the end of the machining step
- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
    - Main drive
    - Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)

> ℹ️ Type of machining for technology database access:
> **Finishing**

Cycle run:

1 Move along a paraxial path from the **Start point** to the **Start point contour X1**, **Z1**
2 Finish the defined contour area, taking the optional contour elements into account
3 Move to the **Tool change point** according to the setting in **G14**

Longitudinal cutting, ICP contour-parallel

▶ Select **Roughing cycles lon/trans**

▶ Select **ICP-long.parallel contour**

The cycle roughs the defined area along contour-parallel paths.

> 🛈 ■ In this cycle, the workpiece will be roughed in a contour-parallel manner, depending on the **Workpiece blank oversize J** and **Type of cut lines H**:
> - ■ **J** = 0: The area defined by **X**, **Z**, and the ICP contour, taking the oversizes into account
> - ■ **J** > 0: The area defined by the ICP contour (plus oversizes) and the **Workpiece blank oversize J**
> ■ The tool plunges with the maximum possible angle, leaving material remaining

---

NOTICE

Danger of collision!

If **Workpiece blank oversize J** > 0, the control will not check whether the programmed **Feed depth P** is possible with the current cutting geometry in the transverse and longitudinal directions. Danger of collision during machining!

▶ Select a **Feed depth P** that fits the current cutting geometry

---

Cycle parameters:
- ■ **X**, **Z**: **Start point**
- ■ **FK**: **No. of ICP finished part** – name of the contour to be machined
- ■ **P**: **Feed depth** (is evaluated depending on **J**)
    - ■ **J** = 0: **P** is the maximum infeed depth.
      The cycle reduces the infeed depth if the programmed infeed is not possible in the transverse or longitudinal direction due to the cutting geometry.
    - ■ **J** > 0: **P** is the infeed depth.
      This infeed is used in the longitudinal and transverse directions.
- ■ **H**: **Type of cut lines** – Machining will be done as follows
    - ■ **0: Constant mach.depth**
    - ■ **1: Equidistant cut lines**
- ■ **I**, **K**: **O-size X** and **Z**
- ■ **J**: **Workpiece blank oversize**
    - ■ **J** = 0: The cycle machines from the current tool position
    - ■ **J** > 0: The cycle machines the area defined by the workpiece blank oversize
- ■ **HR**: **Main machining direction**
- ■ **SX**, **SZ**: **Cutting limit in X** and **Z**
  Further information: "Cutting limits SX, SZ", Page 203

- **G47**: **Safety clearance**
  Further information: "Safety clearance G47", Page 203
- **T**: **Tool number** – turret pocket number
- **G14**: **Tool change point**
  Further information: "Tool change point G14", Page 203
- **ID**: **ID no.**
- **S**: **Cutting speed** or **Constant speed**
- **F**: **Feed per revolution**
- **BP**: **Interval time** – time during which the feed motion will be interrupted
  The chip is broken by the intermittent feed.
- **BF**: **Feed period** – time interval until the next break
  The chip is broken by the intermittent feed.
- **A**: **Start angle** (reference: Z axis; default: parallel to Z axis)
- **W**: **Depart.angle** (reference: Z axis; default: orthogonal to Z axis)
- **XA**, **ZA**: **Start point blank** (Definition of the corner point of the workpiece blank contour—an evaluation will only take place if no workpiece blank has been defined beforehand)
  - **XA**, **ZA** not programmed: The workpiece blank contour is calculated from the tool position and the ICP contour
  - **XA**, **ZA** programmed: Definition of the corner point of the workpiece blank
- **MT**: **M after T**: **M** function that is executed after the **T** tool call
- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step
- **MFE**: **M at end**: **M** function that is executed at the end of the machining step
- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
  - Main drive
  - Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)

> ℹ️ Type of machining for technology database access:
> **Roughing**

Cycle run:

1 Calculate the proportioning of cuts (infeed), taking the **Workpiece blank oversize J** and the **Type of cut lines** into account **H**

- **J** = 0: The cutting geometry is taken into account. This may result in the use of different infeeds for the longitudinal and transverse directions
- **J** > 0: The same infeed is used for both the longitudinal and the transverse directions.

2 Approach the workpiece from the **Start point** along a paraxial path for the first pass

3 Machine the workpiece according to the calculated proportioning of cuts

4 Retract and approach for the next pass

5 Repeat 3 to 4 until the defined area has been machined

6 Return to the **Start point** along a paraxial path

7 Move to the **Tool change point** according to the setting in **G14**

Cutting, transverse, ICP contour-parallel

▶ Select **Roughing cycles lon/trans**

▶ Select **ICP transv. parallel contour**

The cycle roughs the defined area on contour-parallel paths.

🛈 ▪ In this cycle, the workpiece will be roughed in a contour-parallel manner, depending on the **Workpiece blank oversize J** and **Type of cut lines H**:
   ▪ **J** = 0: The area defined by **X**, **Z**, and the ICP contour, taking the oversizes into account
   ▪ **J** > 0: The area defined by the ICP contour (plus oversizes) and the **Workpiece blank oversize J**
   ▪ The tool plunges with the maximum possible angle, leaving material remaining

---

NOTICE

Danger of collision!

If **Workpiece blank oversize J** > 0, the control will not check whether the programmed **Feed depth P** is possible with the current cutting geometry in the transverse and longitudinal directions. Danger of collision during machining!

▶ Select a **Feed depth P** that fits the current cutting geometry

---

Cycle parameters:
- **X**, **Z**: **Start point**
- **FK**: **No. of ICP finished part** – name of the contour to be machined
- **P**: **Feed depth** (is evaluated depending on **J**)
  - **J** = 0: **P** is the maximum infeed depth.
    The cycle reduces the infeed depth if the programmed infeed is not possible in the transverse or longitudinal direction due to the cutting geometry.
  - **J** > 0: **P** is the infeed depth.
    This infeed is used in the longitudinal and transverse directions.
- **H**: **Type of cut lines** – Machining will be done as follows
  - **0: Constant mach.depth**
  - **1: Equidistant cut lines**
- **I**, **K**: **O-size X** and **Z**
- **J**: **Workpiece blank oversize**
  - **J** = 0: The cycle machines from the current tool position
  - **J** > 0: The cycle machines the area defined by the workpiece blank oversize
- **HR**: **Main machining direction**
- **SX**, **SZ**: **Cutting limit in X** and **Z**
  Further information: "Cutting limits SX, SZ", Page 203

- **G47**: **Safety clearance**
  Further information: "Safety clearance G47", Page 203
- **T**: **Tool number** – turret pocket number
- **G14**: **Tool change point**
  Further information: "Tool change point G14", Page 203
- **ID**: **ID no.**
- **S**: **Cutting speed** or **Constant speed**
- **F**: **Feed per revolution**
- **BP**: **Interval time** – time during which the feed motion will be interrupted
  The chip is broken by the intermittent feed.
- **BF**: **Feed period** – time interval until the next break
  The chip is broken by the intermittent feed.
- **A**: **Start angle** (reference: Z axis; default: orthogonal to Z axis)
- **W**: **Depart.angle** (reference: Z axis; default: parallel to Z axis)
- **XA**, **ZA**: **Start point blank** (Definition of the corner point of the workpiece blank contour—an evaluation will only take place if no workpiece blank has been defined beforehand)
  - **XA**, **ZA** not programmed: The workpiece blank contour is calculated from the tool position and the ICP contour
  - **XA**, **ZA** programmed: Definition of the corner point of the workpiece blank
- **MT**: **M after T**: **M** function that is executed after the **T** tool call
- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step
- **MFE**: **M at end**: **M** function that is executed at the end of the machining step
- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
  - Main drive
  - Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)

> **ℹ** Type of machining for technology database access:
> **Roughing**

Cycle run:

1 Calculate the proportioning of cuts (infeed), taking the **Workpiece blank oversize J** and the **Type of cut lines H** into account

- **J** = 0: The cutting geometry is taken into account. This may result in the use of different infeeds for the longitudinal and transverse directions

- **J** > 0: The same infeed is used for both the longitudinal and the transverse directions.

2 Approach the workpiece from the **Start point** on a paraxial path for the first pass

3 Machine the workpiece according to the calculated proportioning of cuts

4 Retract and approach for the next pass

5 Repeat 3 to 4 until the defined area has been machined

6 Return to the **Start point** on a paraxial path

7 Move to the **Tool change point** according to the setting in **G14**

Longitudinal cutting, ICP contour-parallel, finishing

      ▶   Select **Roughing cycles lon/trans**

      ▶   Select **ICP-long.parallel contour**

      ▶   Press the **Finishing run** soft key

The cycle finishes the contour area defined by the ICP contour. When the cycle is completed, the tool remains at the cycle end position.

> ⓘ   The tool plunges with the maximum possible angle, leaving material remaining.

Cycle parameters:

- **X**, **Z**: **Start point**
- **FK**: **No. of ICP finished part** – name of the contour to be machined
- **DXX**: **Add. correction number** (range: 1 to 16) **Further information:** "Additive compensation Dxx", Page 203
- **G58**: **Contour-parallel oversize**
- **DI**, **DK**: **Oversize X** and **Z** paraxial
- **SX**, **SZ**: **Cutting limit in X** and **Z**
  Further information: "Cutting limits SX, SZ", Page 203
- **G47**: **Safety clearance**
  Further information: "Safety clearance G47", Page 203
- **T**: **Tool number** – turret pocket number
- **G14**: **Tool change point**
  Further information: "Tool change point G14", Page 203
- **ID**: **ID no.**
- **S**: **Cutting speed** or **Constant speed**
- **F**: **Feed per revolution**
- **MT**: **M after T**: **M** function that is executed after the **T** tool call
- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step
- **MFE**: **M at end**: **M** function that is executed at the end of the machining step
- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
  - Main drive
  - Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)

> ⓘ   Type of machining for technology database access:
> **Finishing**

Cycle run:

1 Move along a paraxial path from the **Start point** to the ICP contour starting point
2 Finish the defined contour area
3 Move to the **Tool change point** according to the setting in **G14**

Cut, ICP contour-parallel, transverse finishing

▶ Select **Roughing cycles lon/trans**

▶ Select **ICP transv. parallel contour**

Finishing
run

▶ Press the **Finishing run** soft key

The cycle finishes the contour area defined by the ICP contour. When the cycle is completed, the tool remains at the cycle end position.

> ⓘ The tool plunges with the maximum possible angle, leaving material remaining.

Cycle parameters:

- **X**, **Z**: **Start point**
- **FK**: **No. of ICP finished part** – name of the contour to be machined
- **DXX**: **Add. correction number** (range: 1 to 16) **Further information:** "Additive compensation Dxx", Page 203
- **G58**: **Contour-parallel oversize**
- **DI**, **DK**: **Oversize X** and **Z** paraxial
- **SX**, **SZ**: **Cutting limit in X** and **Z** **Further information:** "Cutting limits SX, SZ", Page 203
- **G47**: **Safety clearance** **Further information:** "Safety clearance G47", Page 203
- **T**: **Tool number** – turret pocket number
- **G14**: **Tool change point** **Further information:** "Tool change point G14", Page 203
- **ID**: **ID no.**
- **S**: **Cutting speed** or **Constant speed**
- **F**: **Feed per revolution**
- **MT**: **M after T**: **M** function that is executed after the **T** tool call
- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step
- **MFE**: **M at end**: **M** function that is executed at the end of the machining step
- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
  - Main drive
  - Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)

> ⓘ Type of machining for technology database access: **Finishing**

Cycle run:

1   Move along a paraxial path from the **Start point** to the ICP contour starting point
2   Finish the defined contour area
3   Move to the **Tool change point** according to the setting in **G14**

ICP cut longitud.

▶   Select **Roughing cycles lon/trans**

▶   Select **ICP cutting longitudinal**

The cycle roughs the area defined by the **Start point** and the ICP contour, taking the oversizes into account.
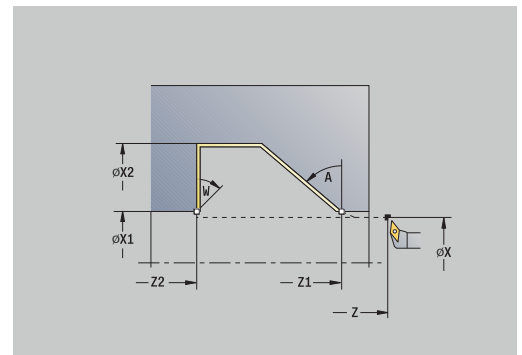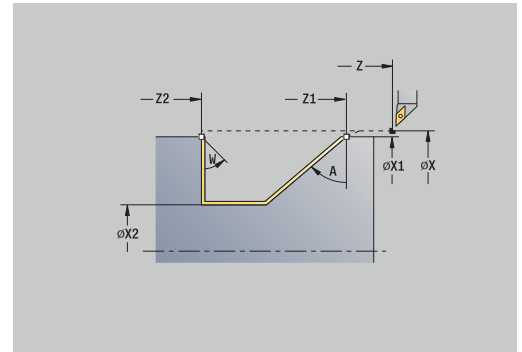
> ■ The tool plunges with the maximum possible angle, leaving material remaining
> ■ The steeper the tool plunges into the material, the greater the feed rate decrease (max. 50 %)

Cycle parameters:
- **X**, **Z**: **Start point**
- **FK**: **No. of ICP finished part** – name of the contour to be machined
- **P**: **Feed depth** – Maximum infeed depth
- **H**: **Contour smoothing**
    - **0: With each cut**
    - **1: With the last cut**
    - **2: No smoothing**
- **I**, **K**: **O-size X** and **Z**
- **E**: **Plunging behavior**
    - No input: Automatic feed-rate reduction
    - **E** = 0: No plunging
    - **E** > 0: Plunging feed rate in use
- **O**: **Hide undercutting**
- **SX**, **SZ**: **Cutting limit in X** and **Z**
    Further information: "Cutting limits SX, SZ", Page 203
- **G47**: **Safety clearance**
    Further information: "Safety clearance G47", Page 203
- **T**: **Tool number** – turret pocket number
- **G14**: **Tool change point**
    Further information: "Tool change point G14", Page 203
- **ID**: **ID no.**
- **S**: **Cutting speed** or **Constant speed**
- **F**: **Feed per revolution**
- **BP**: **Interval time** – time during which the feed motion will be interrupted
    The chip is broken by the intermittent feed.
- **BF**: **Feed period** – time interval until the next break
    The chip is broken by the intermittent feed.
- **A**: **Start angle** (reference: Z axis; default: parallel to Z axis)
- **W**: **Depart.angle** (reference: Z axis; default: orthogonal to Z axis)
- **XA**, **ZA**: **Start point blank** (Definition of the corner point of the workpiece blank contour—an evaluation will only take place if no workpiece blank has been defined beforehand)

- XA, ZA not programmed: The workpiece blank contour is calculated from the tool position and the ICP contour
- XA, ZA programmed: Definition of the corner point of the workpiece blank
- **MT**: **M after T**: **M** function that is executed after the **T** tool call
- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step
- **MFE**: **M at end**: **M** function that is executed at the end of the machining step
- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
  - Main drive
  - Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)

> **ℹ** Type of machining for technology database access: **Roughing**

Cycle run:
1. Calculate the proportioning of cuts (infeed)
2. Approach the workpiece from the **Start point** along a paraxial path for the first pass
3. For descending contours, plunge into the material at a reduced feed rate
4. Machine the workpiece according to the calculated proportioning of cuts
5. The contour is machined according to the setting in **Contour smoothing H**
6. Retract and approach for the next pass
7. Repeat 3 to 6 until the defined area has been machined
8. Return to the **Start point** along a paraxial path
9. Move to the **Tool change point** according to the setting in **G14**

ICP cutting transverse

▶ Select **Roughing cycles lon/trans**

▶ Select **ICP cutting transverse**

The cycle roughs the area defined by the start point and the ICP contour, taking the oversizes into account.
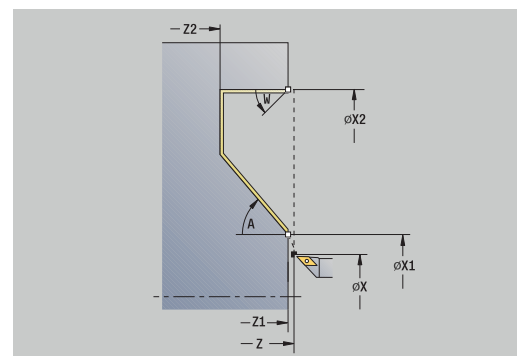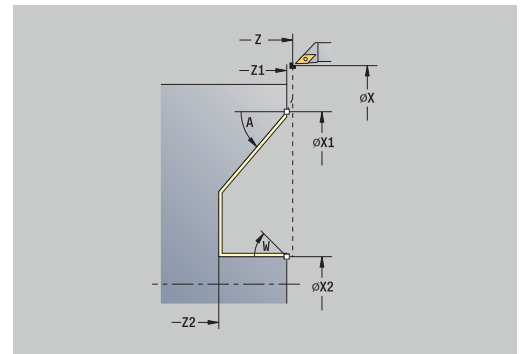
> **ℹ** ▪ The tool plunges with the maximum possible angle, leaving material remaining
> ▪ The steeper the tool plunges into the material, the greater the feed rate decrease (max. 50 %)

Cycle parameters:
- **X**, **Z**: **Start point**
- **FK**: **No. of ICP finished part** – name of the contour to be machined
- **P**: **Feed depth** – Maximum infeed depth
- **H**: **Contour smoothing**
  - **0: With each cut**
  - **1: With the last cut**
  - **2: No smoothing**
- **I**, **K**: **O-size X** and **Z**
- **E**: **Plunging behavior**
  - No input: Automatic feed-rate reduction
  - **E** = 0: No plunging
  - **E** > 0: Plunging feed rate in use
- **O**: **Hide undercutting**
- **SX**, **SZ**: **Cutting limit in X** and **Z**
  Further information: "Cutting limits SX, SZ", Page 203
- **G47**: **Safety clearance**
  Further information: "Safety clearance G47", Page 203
- **T**: **Tool number** – turret pocket number
- **G14**: **Tool change point**
  Further information: "Tool change point G14", Page 203
- **ID**: **ID no.**
- **S**: **Cutting speed** or **Constant speed**
- **F**: **Feed per revolution**
- **BP**: **Interval time** – time during which the feed motion will be interrupted
  The chip is broken by the intermittent feed.
- **BF**: **Feed period** – time interval until the next break
  The chip is broken by the intermittent feed.
- **A**: **Start angle** (reference: Z axis; default: parallel to Z axis)
- **W**: **Depart.angle** (reference: Z axis; default: orthogonal to Z axis)
- **XA**, **ZA**: **Start point blank** (Definition of the corner point of the workpiece blank contour—an evaluation will only take place if no workpiece blank has been defined beforehand)

- **XA**, **ZA** not programmed: The workpiece blank contour is calculated from the tool position and the ICP contour
  - **XA**, **ZA** programmed: Definition of the corner point of the workpiece blank
- **MT**: **M after T**: **M** function that is executed after the **T** tool call
- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step
- **MFE**: **M at end**: **M** function that is executed at the end of the machining step
- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
  - Main drive
  - Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)

> **i** Type of machining for technology database access: **Roughing**

Cycle run:
1 Calculate the proportioning of cuts (infeed)
2 Approach the workpiece from the **Start point** along a paraxial path for the first pass
3 For descending contours, plunge into the material at a reduced feed rate
4 Machine the workpiece according to the calculated proportioning of cuts
5 The contour is machined according to the setting in **Contour smoothing H**
6 Retract and approach for the next pass
7 Repeat 3 to 6 until the defined area has been machined
8 Return to the **Start point** along a paraxial path
9 Move to the **Tool change point** according to the setting in **G14**

Longitudinal ICP cutting, finishing

▶ Select **Roughing cycles lon/trans**

▶ Select **ICP cutting longitudinal**

▶ Press the **Finishing run** soft key

The cycle finishes the contour area defined by the ICP contour. When the cycle is completed, the tool remains at the cycle end position.

> ⓘ The tool plunges with the maximum possible angle, leaving material remaining.

Cycle parameters:
- **X**, **Z**: **Start point**
- **FK**: **No. of ICP finished part** – name of the contour to be machined
- **DXX**: **Add. correction number** (range: 1 to 16) **Further information:** "Additive compensation Dxx", Page 203
- **G58**: **Contour-parallel oversize**
- **DI**, **DK**: **Oversize X** and **Z** paraxial
- **SX**, **SZ**: **Cutting limit in X** and **Z** **Further information:** "Cutting limits SX, SZ", Page 203
- **G47**: **Safety clearance** **Further information:** "Safety clearance G47", Page 203
- **T**: **Tool number** – turret pocket number
- **G14**: **Tool change point** **Further information:** "Tool change point G14", Page 203
- **ID**: **ID no.**
- **S**: **Cutting speed** or **Constant speed**
- **F**: **Feed per revolution**
- **MT**: **M after T**: **M** function that is executed after the **T** tool call
- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step
- **MFE**: **M at end**: **M** function that is executed at the end of the machining step
- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
  - Main drive
  - Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)

> ⓘ Type of machining for technology database access: **Finishing**

Cycle run:

1 Move along a paraxial path from the **Start point** to the ICP contour starting point

2 Finish the defined contour area

3 Move to the **Tool change point** according to the setting in **G14**

Transverse ICP cutting, finishing

▶ Select **Roughing cycles lon/trans**

▶ Select **ICP cutting transverse**

▶ Press the **Finishing run** soft key

The cycle finishes the contour area defined by the ICP contour. When the cycle is completed, the tool remains at the cycle end position.

> ℹ The tool plunges with the maximum possible angle, leaving material remaining.

Cycle parameters:

- **X**, **Z**: **Start point**
- **FK**: **No. of ICP finished part** – name of the contour to be machined
- **DXX**: **Add. correction number** (range: 1 to 16) **Further information:** "Additive compensation Dxx", Page 203
- **G58**: **Contour-parallel oversize**
- **DI**, **DK**: **Oversize X** and **Z** paraxial
- **SX**, **SZ**: **Cutting limit in X** and **Z** **Further information:** "Cutting limits SX, SZ", Page 203
- **G47**: **Safety clearance** **Further information:** "Safety clearance G47", Page 203
- **T**: **Tool number** – turret pocket number
- **G14**: **Tool change point** **Further information:** "Tool change point G14", Page 203
- **ID**: **ID no.**
- **S**: **Cutting speed** or **Constant speed**
- **F**: **Feed per revolution**
- **MT**: **M after T**: **M** function that is executed after the **T** tool call
- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step
- **MFE**: **M at end**: **M** function that is executed at the end of the machining step
- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
  - Main drive
  - Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)

> ℹ Type of machining for technology database access: **Finishing**

Cycle run:

1   Move along a paraxial path from the **Start point** to the ICP contour starting point
2   Finish the defined contour area
3   Move to the **Tool change point** according to the setting in **G14**

Examples of turning cycles

Roughing and finishing an outside contour

The shaded area from **AP** (**Start point contour**) to **EP** (**End point contour**) is roughed with the "Cut longitudinal, expanded" cycle, taking oversizes into account. This contour area will be finished subsequently with the "Finishing cut longitudinal, expanded" cycle.

The rounding arc and the oblique cut at the contour end are both machined in **expanded mode.**

The **Start point contour X1**, **Z1** and **End point contour X2**, **Z2** parameters determine the cutting and infeed directions—in this example, outside machining and infeed in negative X-axis direction.

Tool data:

- Turning tool (for external machining)
- **TO** = 1 – Tool orientation
- **A** = 93° – Tool angle
- **B** = 55° – Point angle
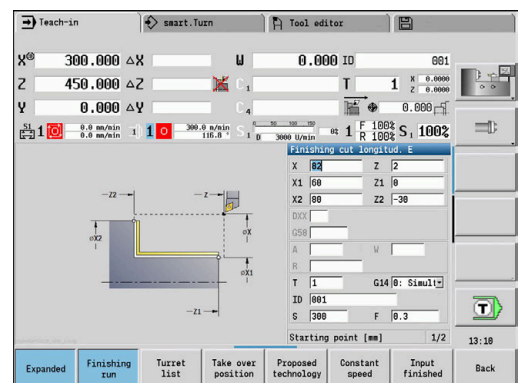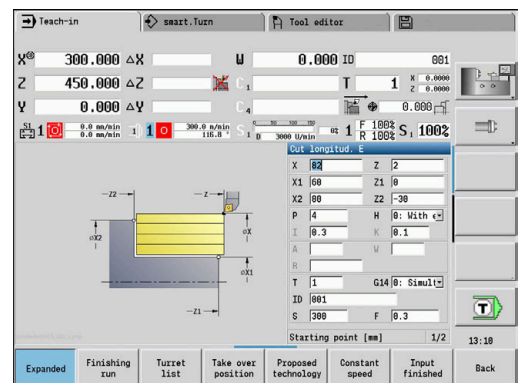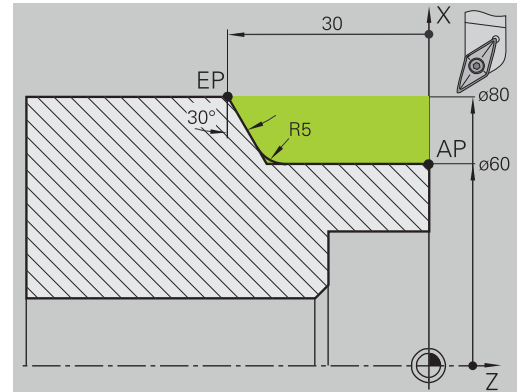
Roughing and finishing an inside contour

The shaded area from **AP** (**Start point contour**) to **EP** (**End point contour**) is roughed with the "Cut longitudinal, expanded" cycle, taking oversizes into account. This contour area will be finished subsequently with the "Finishing cut longitudinal, expanded" cycle.

The rounding arc and the chamfer at the contour end are both machined in **expanded mode.**

The **Start point contour X1**, **Z1** and **End point contour X2**, **Z2** parameters determine the cutting and infeed directions—in this example, inside machining and infeed in positive X-axis direction.

Tool data:

- Turning tool (for internal machining)
- **TO** = 7 – Tool orientation
- **A** = 93° – Tool angle
- **B** = 55° – Point angle
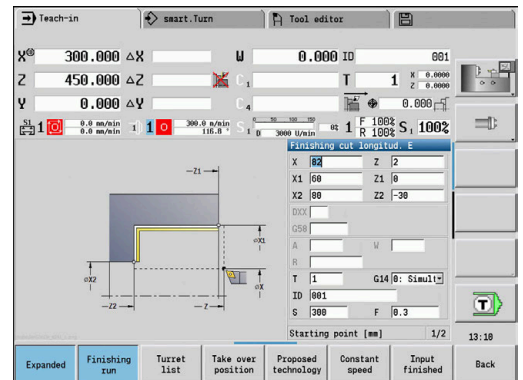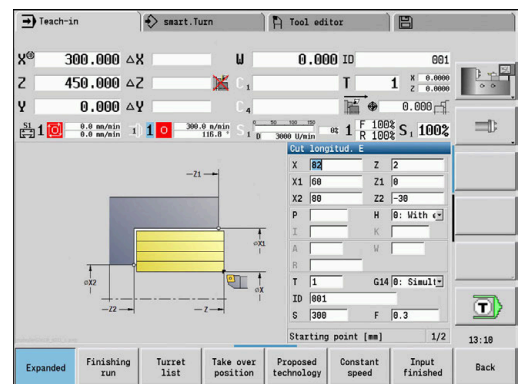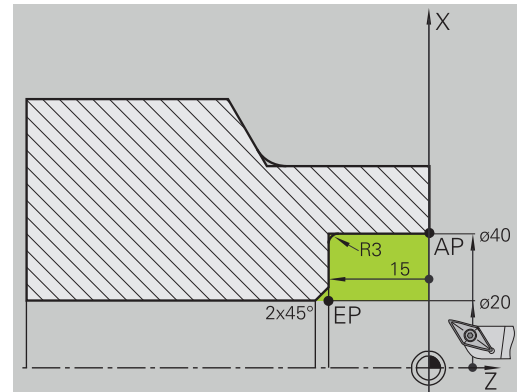
Roughing (hollowing) with the plunge cycle

The tool to be used cannot plunge at the required angle of 15°. The roughing process for the area therefore requires two steps.

1st step

The shaded area from **AP** (**Start point contour**) to **EP** (**End point contour**) is roughed with the **Plunge longit. E** cycle, taking oversizes into account.

The **Start. angle A** is defined with 15°, as dimensioned in the workpiece drawing. Based on the tool parameters, the control calculates the maximum plunge angle that can be achieved with the programmed tool. The residual material will be cut off in the second step.

The **expanded mode** is used to machine the rounding arcs in the contour valley.

Be sure to enter the correct values for the **Start point contour X1**, **Z1** and **End point contour X2**, **Z2** parameters. These parameters determine the cutting and infeed directions—in this example, outside machining and infeed in negative X-axis direction.

Tool data:

- Turning tool (for external machining)
- **TO** = 1 – Tool orientation
- **A** = 93° – Tool angle
- **B** = 55° – Point angle
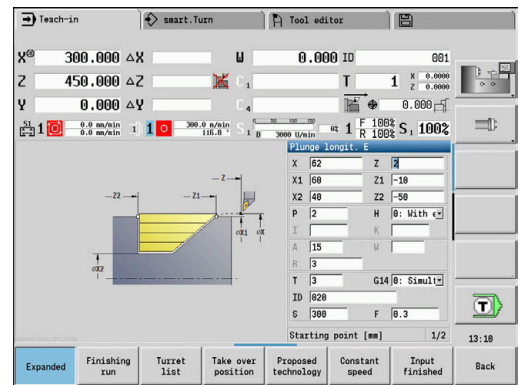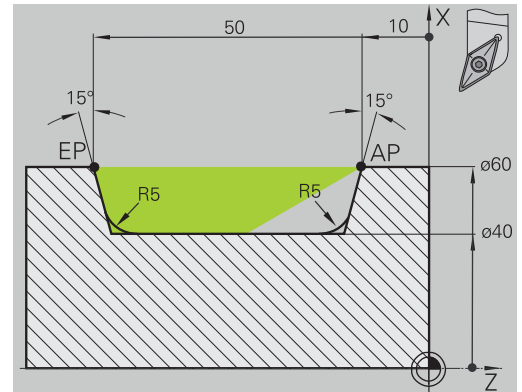
2nd step

The area that was left out in the first step (shaded area in the figure) is machined with the **Plunge longit. E** cycle. Before executing this step, you must change tools.

The **expanded mode** is used to machine the rounding arcs in the contour valley.

The **Start point contour X1**, **Z1** and **End point contour X2**, **Z2** parameters determine the cutting and infeed directions—in this example, outside machining and infeed in negative X-axis direction.

The **Start point contour Z1** parameter was determined during simulation of the first step.

Tool data:

- Turning tool (for external machining)
- **TO** = 3 – Tool orientation
- **A** = 93° – Tool angle
- **B** = 55° – Point angle
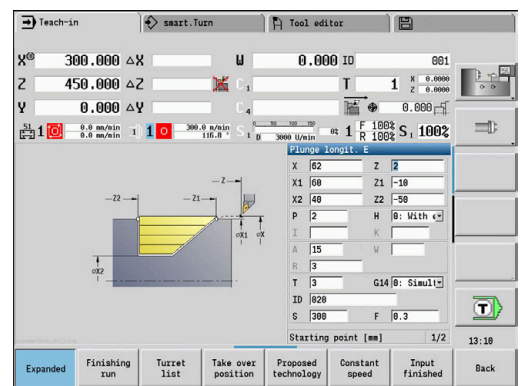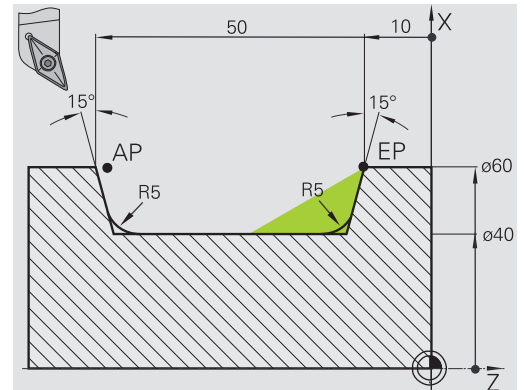
7.5 Recessing cycles

Menu item	Meaning
	The recessing cycle group comprises recessing, recess turning, undercut and parting cycles. Simple contours can be machined in **basic mode,** complex contours in **expanded mode.**

ICP recessing cycles machine contours defined with **ICP**.

Further information: "ICP contours", Page 436

> ℹ️ ■ **Proportioning of cuts:** The control calculates a constant recessing width that is <= **P**
>
> ■ Oversizes are considered in **expanded mode**
>
> ■ Tool-tip radius compensation is conducted (exception: **Undercut type K**)

Cutting and infeed directions for recessing cycles:

The control determines the cutting and infeed directions from the cycle parameters.

The decisive ones are:

■ **Basic mode:** The parameters for **Starting point X**, **Z** (in **Machine** operating mode: current tool position) and **Start point contour X1** / **End point contour Z2**

■ **Expanded mode**: The parameters for **Start point contour X1**, **Z1** and **End point contour X2**, **Z2**

■ **ICP cycles:** The parameters for **Starting point X**, **Z** (in **Machine** operating mode: current tool position) and starting point of ICP contour

Menu item		Recessing cycles
		Recessing radial/Recessing axial
		Recessing and finishing cycles for simple contours
		ICP recessing radial/ICP recessing axial
		Recessing and finishing cycles for any contour
		Recess turning radial/Recess turning axial
		Recess-turning and finishing cycles for simple contours and any type of contour
		Undercut. Form H
		Undercut type H
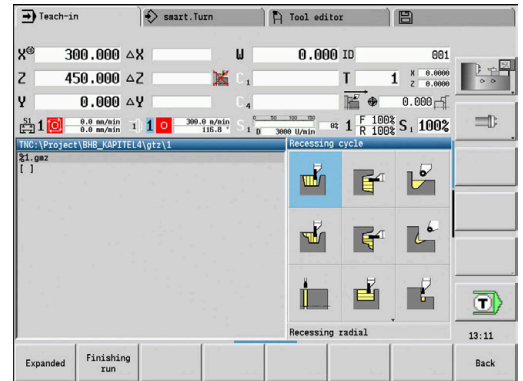
Menu item	Recessing cycles
	Undercut. Form K Undercut type K
	Undercut. Form U Undercut type U
	Cut-off Cycle for parting the workpiece

Undercut position

The control determines the position of an undercut from the cycle parameters for **Start point X**, **Z** (in **Machine** mode: current tool position) and **Start point contour X1**, **Z1**.

Contour forms

Contour elements in recessing cycles

	Basic mode Machining a rectangular area
	Expanded mode Oblique cut at contour start
	Expanded mode Oblique cut at contour end
	Expanded mode Rounding arc in both corners of contour valley
	Expanded mode Chamfer or rounding arc at contour start
	Expanded mode Chamfer or rounding arc at contour end

Recessing radial

▶ Select **Recessing cycle**

▶ Select **Recessing radial**

The cycle machines the number of recesses defined in **Number of recess cycles Qn**. The **Start point** and **End point contour** parameters define the first recess (position, recess depth, and recess width).

Cycle parameters:

- **X**, **Z**: **Start point**
- **X2**, **Z2**: **End point contour**
- **P**: **Cut. width** – Infeeds <= **P** (no input: **P** = 0.8 * cutting width of the tool)
- **EZ**: **Delay** – Dwell time for chip breaking (default: length of time for two revolutions)
- **Qn**: **Number of recess cycles** (default: 1)
- **DX**, **DZ**: **Dist.to subsequent recess** relative to the preceding recess
- **G47**: **Safety clearance**
  Further information: "Safety clearance G47", Page 203
- **T**: **Tool number** – turret pocket number
- **G14**: **Tool change point**
  Further information: "Tool change point G14", Page 203
- **ID**: **ID no.**
- **S**: **Cutting speed** or **Constant speed**
- **F**: **Feed per revolution**
- **MT**: **M after T**: **M** function that is executed after the **T** tool call
- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step
- **MFE**: **M at end**: **M** function that is executed at the end of the machining step
- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
  - Main drive
  - Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)

> **i** Type of machining for technology database access:
> **Contour recessing**

> **ⓘ** In this cycle you can specify how the floor element is machined during the finishing process.
>
> The control evaluates the **recessFinishing** (602414) machining parameter for this purpose. If this parameter is not defined, the floor element is divided in the center.

Cycle run:

1 Calculate the recess positions and the proportioning of cuts
2 Approach the workpiece for the next recess along a paraxial path from the **Start point** or from the last recess
3 Move at programmed feed rate up to **End point contour X2**
4 Remain at this position for the dwell time **Delay EZ**
5 Retract and approach for the next pass
6 Repeat 3 to 5 until the complete recess has been machined
7 Repeat 2 to 6 until all recesses have been machined
8 Return to the **Start point** along a paraxial path
9 Move to the **Tool change point** according to the setting in **G14**

Recessing axial

▶ Select **Recessing cycle**

▶ Select **Recessing axial**

The cycle machines the number of recesses defined in **Number of recess cycles Qn**. The **Start point** and **End point contour** parameters define the first recess (position, recess depth, and recess width).

Cycle parameters:

- **X**, **Z**: **Start point**
- **X2**, **Z2**: **End point contour**
- **P**: **Cut. width** – Infeeds <= **P** (no input: **P** = 0.8 * cutting width of the tool)
- **EZ**: **Delay** – Dwell time for chip breaking (default: length of time for two revolutions)
- **Qn**: **Number of recess cycles** (default: 1)
- **DX**, **DZ**: **Dist.to subsequent recess** relative to the preceding recess
- **G47**: **Safety clearance**
  Further information: "Safety clearance G47", Page 203
- **T**: **Tool number** – turret pocket number
- **G14**: **Tool change point**
  Further information: "Tool change point G14", Page 203
- **ID**: **ID no.**
- **S**: **Cutting speed** or **Constant speed**
- **F**: **Feed per revolution**
- **MT**: **M after T**: **M** function that is executed after the **T** tool call
- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step
- **MFE**: **M at end**: **M** function that is executed at the end of the machining step
- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
  - Main drive
  - Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)

> **ⓘ** Type of machining for technology database access:
> **Contour recessing**

> **ℹ** In this cycle you can specify how the floor element is machined during the finishing process.
>
> The control evaluates the **recessFinishing** (602414) machining parameter for this purpose. If this parameter is not defined, the floor element is divided in the center.

Cycle run:

1 Calculate the recess positions and the proportioning of cuts
2 Approach the workpiece for the next recess on a paraxial path from the **Start point** or from the last recess
3 Move to the **End point contour Z2** at the programmed feed rate
4 Remain at this position for the dwell time **Delay EZ**
5 Retract and approach for the next pass
6 Repeat 3 to 5 until the complete recess has been machined
7 Repeat 2 to 6 until all recesses have been machined
8 Return to the **Start point** on a paraxial path
9 Move to the **Tool change point** according to the setting in **G14**

Recessing radial – Expanded

▶ Select **Recessing cycle**

▶ Select **Recessing radial**

▶ Select the **Expanded** soft key

The cycle machines the number of recesses defined in **Number of recess cycles Qn**. The **Start point** and **End point contour** parameters define the first recess (position, recess depth, and recess width).

Cycle parameters:

- **X**, **Z**: **Start point**
- **X1**, **Z1**: **Start point contour**
- **X2**, **Z2**: **End point contour**
- **B1**, **B2**: **-B cham./+B round.** (**B1** at contour start and **B2** at contour end)
    - **B** > 0: Radius of rounding
    - **B** < 0: Width of chamfer
- **A**: **Start angle** (range: 0° <= **A** < 90°; default: 0°)
- **W**: **Final angle** (range: 0° <= **W** < 90°; default: 0°)
- **R**: **Rounding**
- **I**, **K**: **O-size X** and **Z**
- **T**: **Tool number** – turret pocket number
- **G14**: **Tool change point**
  Further information: "Tool change point G14", Page 203
- **ID**: **ID no.**
- **S**: **Cutting speed** or **Constant speed**
- **F**: **Feed per revolution**
- **P**: **Cut. width** – Infeeds <= **P** (no input: **P** = 0.8 * cutting width of the tool)
- **ET**: **Recess depth** per infeed
- **EZ**: **Delay** – Dwell time for chip breaking (default: length of time for two revolutions)
- **EW**: **Recessing feed**
- **Qn**: **Number of recess cycles** (default: 1)
- **DX**, **DZ**: **Dist.to subsequent recess** relative to the preceding recess
- **G47**: **Safety clearance**
  Further information: "Safety clearance G47", Page 203
- **KS**: **Multiple plunging** (default: 0)
    - 0: No
    - 1: Yes – Rough grooving is performed by means of full-section cuts—the ridges in between are machined centrally relative to the recessing tool
- **MT**: **M after T**: **M** function that is executed after the **T** tool call
- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step

- **MFE**: **M at end**: **M** function that is executed at the end of the machining step
- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
  - Main drive
  - Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)

> **i** Type of machining for technology database access:
> **Contour recessing**

> **i** In this cycle you can specify how the floor element is machined during the finishing process.
>
> The control evaluates the **recessFinishing** (602414) machining parameter for this purpose. If this parameter is not defined, the floor element is divided in the center.

Cycle run:

1. Calculate the recess positions and the proportioning of cuts
2. Approach the workpiece for the next recess along a paraxial path from the **Start point** or from the last recess
3. Move at the programmed feed rate to the **End point contour X2** or, if defined, to one of the optional contour elements
4. Remain at this position for the dwell time **Delay EZ**
5. Retract and approach for the next pass
6. Repeat 3 to 5 until the complete recess has been machined
7. Repeat 2 to 6 until all recesses have been machined
8. Return to the **Start point** along a paraxial path
9. Move to the **Tool change point** according to the setting in **G14**

Recessing axial – Expanded

▶ Select **Recessing cycle**

▶ Select **Recessing axial**

Expanded

▶ Select the **Expanded** soft key

The cycle machines the number of recesses defined in **Number of recess cycles Qn**. The **Start point** and **End point contour** parameters define the first recess (position, recess depth, and recess width).

Cycle parameters:
- **X**, **Z**: **Start point**
- **X1**, **Z1**: **Start point contour**
- **X2**, **Z2**: **End point contour**
- **B1**, **B2**: **-B cham./+B round.** (**B1** at contour start and **B2** at contour end)
    - **B** > 0: Radius of rounding
    - **B** < 0: Width of chamfer
- **A**: **Start angle** (range: 0° <= **A** < 90°; default: 0°)
- **W**: **Final angle** (range: 0° <= **W** < 90°; default: 0°)
- **R**: **Rounding**
- **I**, **K**: **O-size X** and **Z**
- **T**: **Tool number** – turret pocket number
- **G14**: **Tool change point**
  Further information: "Tool change point G14", Page 203
- **ID**: **ID no.**
- **S**: **Cutting speed** or **Constant speed**
- **F**: **Feed per revolution**
- **P**: **Cut. width** – Infeeds <= **P** (no input: **P** = 0.8 * cutting width of the tool)
- **ET**: **Recess depth** per infeed
- **EZ**: **Delay** – Dwell time for chip breaking (default: length of time for two revolutions)
- **EW**: **Recessing feed**
- **Qn**: **Number of recess cycles** (default: 1)
- **DX**, **DZ**: **Dist.to subsequent recess** relative to the preceding recess
- **G47**: **Safety clearance**
  Further information: "Safety clearance G47", Page 203
- **KS**: **Multiple plunging** (default: 0)
    - 0: No
    - 1: Yes – Rough grooving is performed by means of full-section cuts—the ridges in between are machined centrally relative to the recessing tool
- **MT**: **M after T**: **M** function that is executed after the **T** tool call
- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step

- **MFE**: **M at end**: **M** function that is executed at the end of the machining step
- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
  - Main drive
  - Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)

> **i** Type of machining for technology database access:
> **Contour recessing**

> **i** In this cycle you can specify how the floor element is machined during the finishing process.
>
> The control evaluates the **recessFinishing** (602414) machining parameter for this purpose. If this parameter is not defined, the floor element is divided in the center.

Cycle run:

1. Calculate the recess positions and the proportioning of cuts
2. Approach the workpiece for the next recess along a paraxial path from the **Start point** or from the last recess
3. Move at the programmed feed rate to the **End point contour Z2** or, if defined, to one of the optional contour elements
4. Remain at this position for the dwell time **Delay EZ**
5. Retract and approach for the next pass
6. Repeat 3 to 5 until the complete recess has been machined
7. Repeat 2 to 6 until all recesses have been machined
8. Return to the **Start point** along a paraxial path
9. Move to the **Tool change point** according to the setting in **G14**

Recessing radial finishing

▶ Select **Recessing cycle**

▶ Select **Recessing radial**

▶ Press the **Finishing run** soft key

The cycle machines the number of recesses defined in **Number of recess cycles Qn**. The **Start point** and **End point contour** parameters define the first recess (position, recess depth, and recess width).

Cycle parameters:

- **X**, **Z**: **Start point**
- **X2**, **Z2**: **End point contour**
- **Qn**: **Number of recess cycles** (default: 1)
- **DX**, **DZ**: **Dist.to subsequent recess** relative to the preceding recess
- **G47**: **Safety clearance**
  Further information: "Safety clearance G47", Page 203
- **T**: **Tool number** – turret pocket number
- **G14**: **Tool change point**
  Further information: "Tool change point G14", Page 203
- **ID**: **ID no.**
- **S**: **Cutting speed** or **Constant speed**
- **F**: **Feed per revolution**
- **MT**: **M after T**: **M** function that is executed after the **T** tool call
- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step
- **MFE**: **M at end**: **M** function that is executed at the end of the machining step
- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
  - Main drive
  - Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)

> **i** Type of machining for technology database access:
> **Contour recessing**

> **i** In this cycle you can specify how the floor element is machined during the finishing process.
> The control evaluates the **recessFinishing** (602414) machining parameter for this purpose. If this parameter is not defined, the floor element is divided in the center.

Cycle run:

1 Calculate the recess positions

2 Approach the workpiece for the next recess along a paraxial path from the **Start point** or from the last recess

3 Finish the first side and the contour valley up to a position just before the recess end point

4 Approach the workpiece along a paraxial path for finishing the second side

5 Finish the second side and the remainder of the contour valley

6 Repeat 2 to 5 until all recesses have been machined

7 Return to the **Start point** along a paraxial path

8 Move to the **Tool change point** according to the setting in **G14**

Recessing axial finishing

▶ Select **Recessing cycle**

▶ Select **Recessing axial**

▶ Press the **Finishing run** soft key

The cycle machines the number of recesses defined in **Number of recess cycles Qn**. The **Start point** and **End point contour** parameters define the first recess (position, recess depth, and recess width).

Cycle parameters:

- **X**, **Z**: **Start point**
- **X2**, **Z2**: **End point contour**
- **Qn**: **Number of recess cycles** (default: 1)
- **DX**, **DZ**: **Dist.to subsequent recess** relative to the preceding recess
- **G47**: **Safety clearance**
  Further information: "Safety clearance G47", Page 203
- **T**: **Tool number** – turret pocket number
- **G14**: **Tool change point**
  Further information: "Tool change point G14", Page 203
- **ID**: **ID no.**
- **S**: **Cutting speed** or **Constant speed**
- **F**: **Feed per revolution**
- **MT**: **M after T**: **M** function that is executed after the **T** tool call
- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step
- **MFE**: **M at end**: **M** function that is executed at the end of the machining step
- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
  - Main drive
  - Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)

> ⓘ Type of machining for technology database access:
> **Contour recessing**

> ⓘ In this cycle you can specify how the floor element is machined during the finishing process.
>
> The control evaluates the **recessFinishing** (602414) machining parameter for this purpose. If this parameter is not defined, the floor element is divided in the center.

Cycle run:

1 Calculate the recess positions
2 Approach the workpiece for the next recess on a paraxial path from the **Start point** or from the last recess
3 Finish the first side and the contour valley up to a position just before the recess end point
4 Approach the workpiece on a paraxial path for finishing the second side
5 Finish the second side and the remainder of the contour valley
6 Repeat 2 to 5 until all recesses have been machined
7 Return to the **Start point** on a paraxial path
8 Move to the **Tool change point** according to the setting in **G14**

Recessing radial finishing – Expanded

▶ Select **Recessing cycle**

▶ Select **Recessing radial**

▶ Select the **Expanded** soft key

▶ Press the **Finishing run** soft key

The cycle machines the number of recesses defined in **Number of recess cycles Qn**. The **Start point** and **End point contour** parameters define the first recess (position, recess depth, and recess width).

Cycle parameters:

- **X**, **Z**: **Start point**
- **X1**, **Z1**: **Start point contour**
- **X2**, **Z2**: **End point contour**
- **B1**, **B2**: **-B cham./+B round.** (**B1** at contour start and **B2** at contour end)
    - **B** > 0: Radius of rounding
    - **B** < 0: Width of chamfer
- **A**: **Start angle** (range: 0° <= **A** < 90°; default: 0°)
- **W**: **Final angle** (range: 0° <= **W** < 90°; default: 0°)
- **R**: **Rounding**
- **T**: **Tool number** – turret pocket number
- **G14**: **Tool change point**
  Further information: "Tool change point G14", Page 203
- **ID**: **ID no.**
- **S**: **Cutting speed** or **Constant speed**
- **F**: **Feed per revolution**
- **Qn**: **Number of recess cycles** (default: 1)
- **DX**, **DZ**: **Dist.to subsequent recess** relative to the preceding recess
- **G47**: **Safety clearance**
  Further information: "Safety clearance G47", Page 203
- **MT**: **M after T**: **M** function that is executed after the **T** tool call
- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step
- **MFE**: **M at end**: **M** function that is executed at the end of the machining step
- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
    - Main drive
    - Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)

> **i** Type of machining for technology database access:
> **Contour recessing**

> **i** In this cycle you can specify how the floor element is machined during the finishing process.
>
> The control evaluates the **recessFinishing** (602414) machining parameter for this purpose. If this parameter is not defined, the floor element is divided in the center.

Cycle run:

1 Calculate the recess positions and the proportioning of cuts
2 Approach the workpiece for the next recess along a paraxial path from the **Start point** or from the last recess
3 Finish the first side (taking the optional contour elements into account) and the contour valley up to a position just before the recess end point
4 Approach the workpiece along a paraxial path for finishing the second side
5 Finish the second side (taking the optional contour elements into account) and the remainder of the contour valley
6 Repeat 2 to 5 until all recesses have been finished
7 Return to the **Start point** along a paraxial path
8 Move to the **Tool change point** according to the setting in **G14**

Recessing axial finishing – Expanded

▶ Select **Recessing cycle**

▶ Select **Recessing axial**

▶ Select the **Expanded** soft key

▶ Press the **Finishing run** soft key

The cycle machines the number of recesses defined in **Number of recess cycles Qn**. The **Start point** and **End point contour** parameters define the first recess (position, recess depth, and recess width).
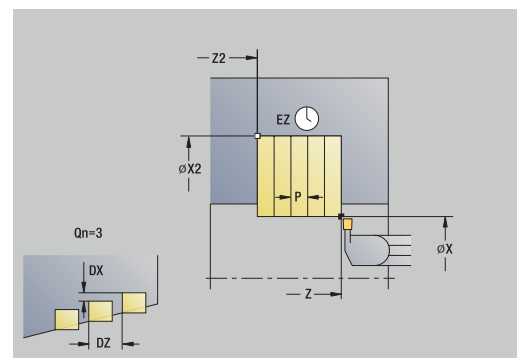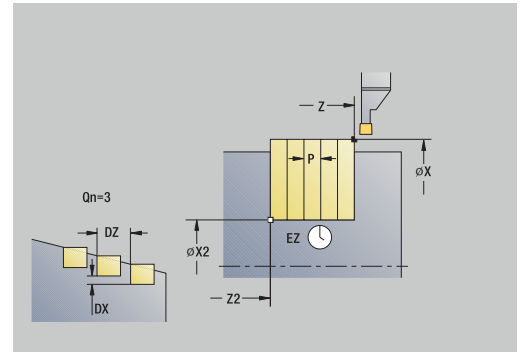
Cycle parameters:

- **X**, **Z**: **Start point**
- **X1**, **Z1**: **Start point contour**
- **X2**, **Z2**: **End point contour**
- **B1**, **B2**: **-B cham./+B round.** (**B1** at contour start and **B2** at contour end)
    - **B** > 0: Radius of rounding
    - **B** < 0: Width of chamfer
- **A**: **Start angle** (range: 0° <= **A** < 90°; default: 0°)
- **W**: **Final angle** (range: 0° <= **W** < 90°; default: 0°)
- **R**: **Rounding**
- **T**: **Tool number** – turret pocket number
- **G14**: **Tool change point**
  Further information: "Tool change point G14", Page 203
- **ID**: **ID no.**
- **S**: **Cutting speed** or **Constant speed**
- **F**: **Feed per revolution**
- **Qn**: **Number of recess cycles** (default: 1)
- **DX**, **DZ**: **Dist.to subsequent recess** relative to the preceding recess
- **G47**: **Safety clearance**
  Further information: "Safety clearance G47", Page 203
- **MT**: **M after T**: **M** function that is executed after the **T** tool call
- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step
- **MFE**: **M at end**: **M** function that is executed at the end of the machining step
- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
    - Main drive
    - Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)

> **ⓘ** Type of machining for technology database access:
> **Contour recessing**

> **ⓘ** In this cycle you can specify how the floor element is machined during the finishing process.
>
> The control evaluates the **recessFinishing** (602414) machining parameter for this purpose. If this parameter is not defined, the floor element is divided in the center.

Cycle run:

1 Calculate the recess positions and the proportioning of cuts
2 Approach the workpiece for the next recess along a paraxial path from the **Start point** or from the last recess
3 Finish the first side (taking the optional contour elements into account) and the contour valley up to a position just before the recess end point
4 Approach the workpiece along a paraxial path for finishing the second side
5 Finish the second side (taking the optional contour elements into account) and the remainder of the contour valley
6 Repeat 2 to 5 until all recesses have been finished
7 Return to the **Start point** along a paraxial path
8 Move to the **Tool change point** according to the setting in **G14**

Radial ICP recessing cycles
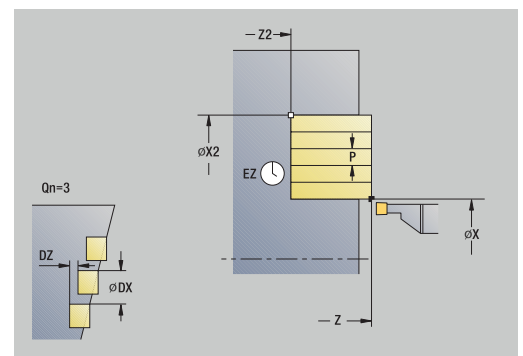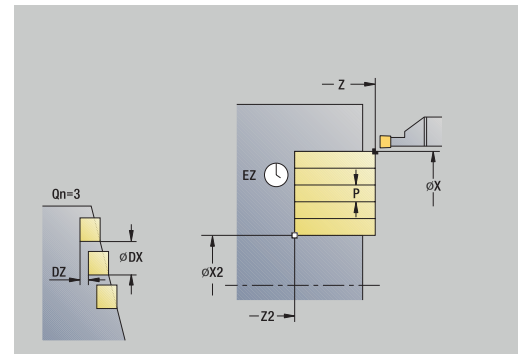

▶ Select **Recessing cycle**

▶ Select **Recessing radial ICP**

The cycle machines the number of recesses defined in **Number of recess cycles Qn**. The **Start point** and **End point contour** parameters define the first recess (position, recess depth, and recess width).

Cycle parameters:

- **X**, **Z**: **Start point**
- **FK**: **No. of ICP finished part** – name of the contour to be machined
- **P**: **Cut. width** – Infeeds <= **P** (no input: **P** = 0.8 * cutting width of the tool)
- **ET**: **Recess depth** per infeed
- **I**, **K**: **O-size X** and **Z**
- **EZ**: **Delay** – Dwell time for chip breaking (default: length of time for two revolutions)
- **EW**: **Recessing feed**
- **Qn**: **Number of recess cycles** (default: 1)
- **DX**, **DZ**: **Dist.to subsequent recess** relative to the preceding recess
- **T**: **Tool number** – turret pocket number
- **G14**: **Tool change point**
  Further information: "Tool change point G14", Page 203
- **ID**: **ID no.**
- **S**: **Cutting speed** or **Constant speed**
- **F**: **Feed per revolution**
- **SX**, **SZ**: **Cutting limit in X** and **Z**
  Further information: "Cutting limits SX, SZ", Page 203
- **G47**: **Safety clearance**
  Further information: "Safety clearance G47", Page 203
- **KS**: **Multiple plunging** (default: 0)
  - 0: No
  - 1: Yes – Rough grooving is performed by means of full-section cuts—the ridges in between are machined centrally relative to the recessing tool
- **MT**: **M after T**: **M** function that is executed after the **T** tool call
- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step
- **MFE**: **M at end**: **M** function that is executed at the end of the machining step
- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
  - Main drive
  - Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)

- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)

> **ℹ** Type of machining for technology database access:
> **Contour recessing**

> **ℹ** In this cycle you can specify how the floor element is machined during the finishing process.
>
> The control evaluates the **recessFinishing** (602414) machining parameter for this purpose. If this parameter is not defined, the floor element is divided in the center.

Cycle run:

1 Calculate the recess positions and the proportioning of cuts
2 Approach the workpiece for the next recess along a paraxial path from the **Start point** or from the last recess
3 Machine along the defined contour
4 Retract and approach for the next pass
5 Repeat 3 to 4 until the complete recess has been machined
6 Repeat 2 to 5 until all recesses have been machined
7 Return to the **Start point** along a paraxial path
8 Move to the **Tool change point** according to the setting in **G14**

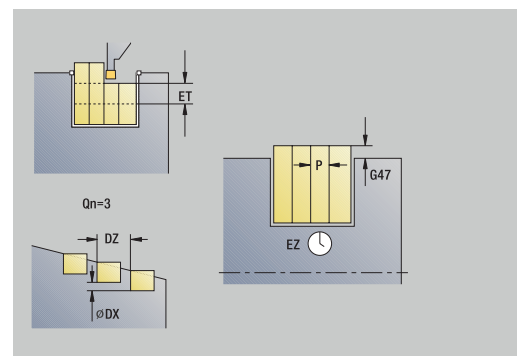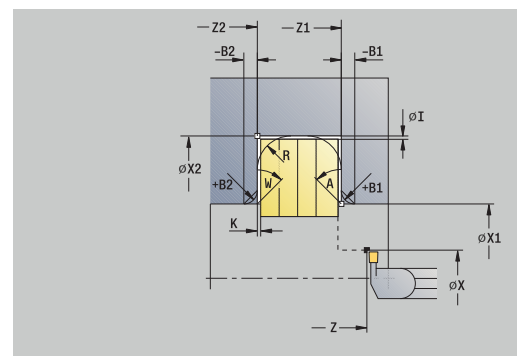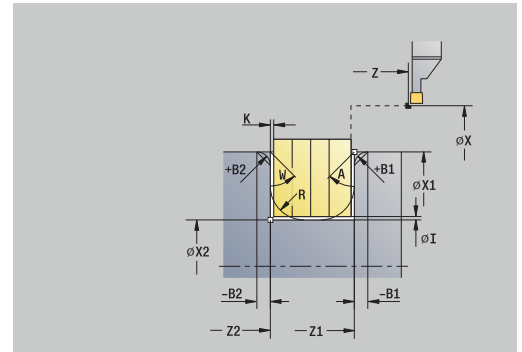## Axial ICP recessing cycles

> ▶ Select **Recessing cycle**

> ▶ Select **Recessing axial ICP**

The cycle machines the number of recesses defined in **Number of recess cycles Qn**. The **Start point** and **End point contour** parameters define the first recess (position, recess depth, and recess width).

Cycle parameters:

- **X**, **Z**: **Start point**
- **FK**: **No. of ICP finished part** – name of the contour to be machined
- **P**: **Cut. width** – Infeeds <= **P** (no input: **P** = 0.8 * cutting width of the tool)
- **ET**: **Recess depth** per infeed
- **I**, **K**: **O-size X** and **Z**
- **EZ**: **Delay** – Dwell time for chip breaking (default: length of time for two revolutions)
- **EW**: **Recessing feed**
- **Qn**: **Number of recess cycles** (default: 1)
- **DX**, **DZ**: **Dist.to subsequent recess** relative to the preceding recess
- **T**: **Tool number** – turret pocket number
- **G14**: **Tool change point**
  Further information: "Tool change point G14", Page 203
- **ID**: **ID no.**
- **S**: **Cutting speed** or **Constant speed**
- **F**: **Feed per revolution**
- **SX**, **SZ**: **Cutting limit in X** and **Z**
  Further information: "Cutting limits SX, SZ", Page 203
- **G47**: **Safety clearance**
  Further information: "Safety clearance G47", Page 203
- **KS**: **Multiple plunging** (default: 0)
  - 0: No
  - 1: Yes – Rough grooving is performed by means of full-section cuts—the ridges in between are machined centrally relative to the recessing tool
- **MT**: **M after T**: **M** function that is executed after the **T** tool call
- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step
- **MFE**: **M at end**: **M** function that is executed at the end of the machining step
- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
  - Main drive
  - Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)

■ **HC**: **Shoe brake** (machine-dependent)

■ **DF**: **Miscellaneous function** (machine-dependent)

> **ⓘ** Type of machining for technology database access:
> **Contour recessing**

> **ⓘ** In this cycle you can specify how the floor element is machined during the finishing process.
>
> The control evaluates the **recessFinishing** (602414) machining parameter for this purpose. If this parameter is not defined, the floor element is divided in the center.

Cycle run:

1 Calculate the recess positions and the proportioning of cuts
2 Approach the workpiece for the next recess on a paraxial path from the **Start point** or from the last recess
3 Machine along the defined contour
4 Return and approach for the next pass
5 Repeat 3 to 4 until the complete recess has been machined
6 Repeat 2 to 5 until all recesses have been machined
7 Return to the **Start point** on a paraxial path
8 Move to the **Tool change point** according to the setting in **G14**

Radial ICP recessing, finishing

▶ Select **Recessing cycle**

▶ Select **Recessing radial ICP**

▶ Press the **Finishing run** soft key
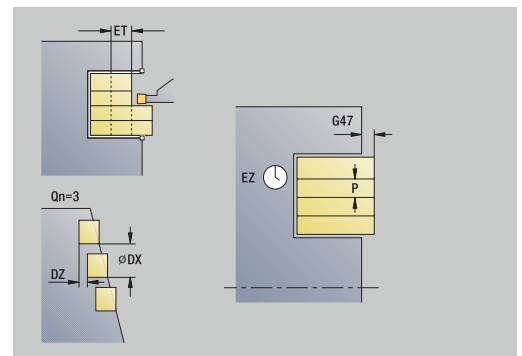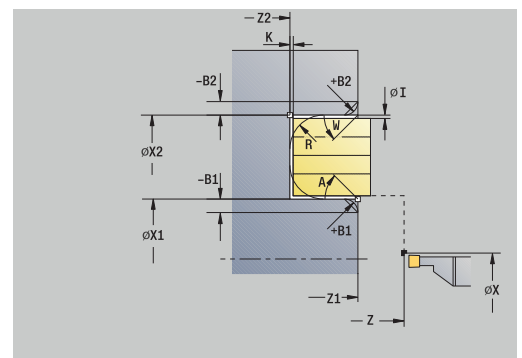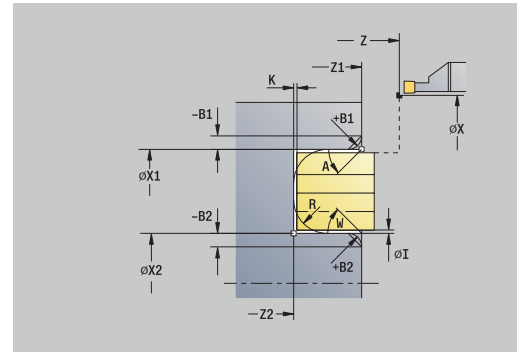
The cycle machines the number of recesses defined in **Number of recess cycles Qn**. The **Start point** and **End point contour** parameters define the first recess (position, recess depth, and recess width).

ℹ At the end of the cycle, the tool returns to the **Start point**.

Cycle parameters:
- **X**, **Z**: **Start point**
- **FK**: **No. of ICP finished part** – name of the contour to be machined
- **Qn**: **Number of recess cycles** (default: 1)
- **DX**, **DZ**: **Dist.to subsequent recess** relative to the preceding recess
- **T**: **Tool number** – turret pocket number
- **G14**: **Tool change point**
  Further information: "Tool change point G14", Page 203
- **ID**: **ID no.**
- **S**: **Cutting speed** or **Constant speed**
- **F**: **Feed per revolution**
- **SX**, **SZ**: **Cutting limit in X** and **Z**
  Further information: "Cutting limits SX, SZ", Page 203
- **G47**: **Safety clearance**
  Further information: "Safety clearance G47", Page 203
- **MT**: **M after T**: **M** function that is executed after the **T** tool call
- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step
- **MFE**: **M at end**: **M** function that is executed at the end of the machining step
- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
  - Main drive
  - Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)

ℹ Type of machining for technology database access:
Contour recessing

> **i** In this cycle you can specify how the floor element is machined during the finishing process.
>
> The control evaluates the **recessFinishing** (602414) machining parameter for this purpose. If this parameter is not defined, the floor element is divided in the center.

Cycle run:

1 Calculate the recess positions
2 Approach the workpiece for the next recess along a paraxial path from the **Start point** or from the last recess
3 Finish the recess
4 Repeat 2 to 3 until all recesses have been machined
5 Return to the **Start point** along a paraxial path
6 Move to the **Tool change point** according to the setting in **G14**

Axial ICP recessing, finishing

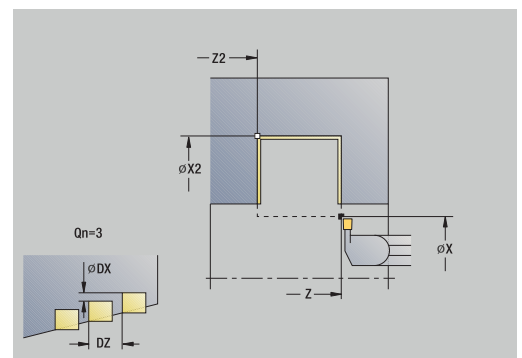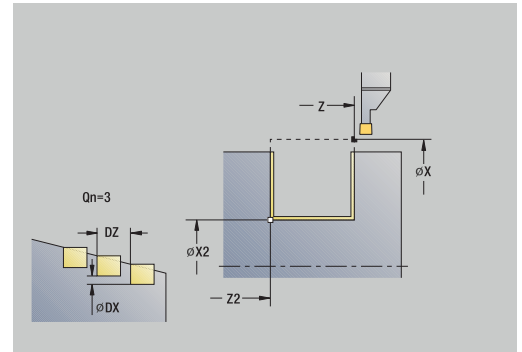▶ Select **Recessing cycle**

▶ Select **Recessing axial ICP**

▶ Press the **Finishing run** soft key

The cycle machines the number of recesses defined in **Number of recess cycles Qn**. The **Start point** and **End point contour** parameters define the first recess (position, recess depth, and recess width).

> **ℹ** At the end of the cycle, the tool returns to the **Start point**.

Cycle parameters:
- **X**, **Z**: **Start point**
- **FK**: **No. of ICP finished part** – name of the contour to be machined
- **Qn**: **Number of recess cycles** (default: 1)
- **DX**, **DZ**: **Dist.to subsequent recess** relative to the preceding recess
- **T**: **Tool number** – turret pocket number
- **G14**: **Tool change point**
  Further information: "Tool change point G14", Page 203
- **ID**: **ID no.**
- **S**: **Cutting speed** or **Constant speed**
- **F**: **Feed per revolution**
- **SX**, **SZ**: **Cutting limit in X** and **Z**
  Further information: "Cutting limits SX, SZ", Page 203
- **G47**: **Safety clearance**
  Further information: "Safety clearance G47", Page 203
- **MT**: **M after T**: **M** function that is executed after the **T** tool call
- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step
- **MFE**: **M at end**: **M** function that is executed at the end of the machining step
- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
  - Main drive
  - Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)

> **ℹ** Type of machining for technology database access:
> **Contour recessing**

🛈 In this cycle you can specify how the floor element is machined during the finishing process.

The control evaluates the **recessFinishing** (602414) machining parameter for this purpose. If this parameter is not defined, the floor element is divided in the center.

Cycle run:

1 Calculate the recess positions
2 Approach the workpiece for the next recess on a paraxial path from the **Start point** or from the last recess
3 Finish the recess
4 Repeat 2 to 3 until all recesses have been machined
5 Return to the **Start point** on a paraxial path
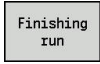6 Move to the **Tool change point** according to the setting in **G14**

Recess turning

The recess turning cycles machine by alternate recessing and roughing movements. The machining process requires a minimum of retraction and infeed movements.

To influence recess-turning operations, use the following parameters:

- **O**: **Recessing feed** – Feed rate for recessing movement
- **U**: **Unidir. turning** – You can perform a unidirectional or bidirectional turning operation
- **B**: **Offset width** – After the second infeed movement, during the transition from turning to recessing, the path to be machined is reduced by the **Offset width**. Each time the system switches from turning to recessing on this side, the path is reduced by the offset width—in addition to the previous offset. The total offset is limited to 80 % of the effective cutting width (effective cutting width = cutting width – 2*cutting radius). If required, the control reduces the programmed **Offset width**. At the end of rough grooving, the residual material is removed with a single cut
- **RB**: **Depth compens.** – Depending on factors such as workpiece material or feed rate the tool tip is displaced during a turning operation. The resulting infeed error can be compensated with **Depth compens.** during **finishing, expanded**. The **Depth compens.** factor is usually determined empirically

🛈 These cycles require the use of **recess-turning tools.**

Radial recess turning

▶ Select **Recessing cycle**

▶ Select **Recess turning**

▶ Select **Radial recess turning**

The cycle machines the rectangle described by the **Start point** and the **End point contour**.

Cycle parameters:

- **X**, **Z**: **Start point**
- **X2**, **Z2**: **End point contour**
- **P**: **Feed depth** – Maximum infeed depth
- **O**: **Recessing feed** (default: active feed rate)
- **B**: **Offset width** (default: 0)
- **U**: **Unidir. turning** (default: 0)
    - **0: Bidirectional**
    - **1: Unidirectional**
- **G47**: **Safety clearance**
  Further information: "Safety clearance G47", Page 203
- **T**: **Tool number** – turret pocket number
- **G14**: **Tool change point**
  Further information: "Tool change point G14", Page 203
- **ID**: **ID no.**
- **S**: **Cutting speed** or **Constant speed**
- **F**: **Feed per revolution**
- **MT**: **M after T**: **M** function that is executed after the **T** tool call
- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step
- **MFE**: **M at end**: **M** function that is executed at the end of the machining step
- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
    - Main drive
    - Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)

> ⓘ   Type of machining for technology database access: **Recess turning**

Cycle run:

1 Calculate the proportioning of cuts
2 Approach the workpiece from the **Start point** for the first pass
3 Execute the first cut (recessing)
4 Machine perpendicularly to the recessing direction (turning)
5 Repeat 3 to 4 until the **End point contour X2**, **Z2** is reached
6 Return to the **Start point** along a paraxial path
7 Move to the **Tool change point** according to the setting in **G14**

Axial recess turning

▶ Select **Recessing cycle**

▶ Select **Recess turning**

▶ Select **Axial recess turning**

The cycle machines the rectangle described by the **Start point** and the **End point contour**.

Cycle parameters:

- **X**, **Z**: **Start point**
- **X2**, **Z2**: **End point contour**
- **P**: **Feed depth** – Maximum infeed depth
- **O**: **Recessing feed** (default: active feed rate)
- **B**: **Offset width** (default: 0)
- **U**: **Unidir. turning** (default: 0)
  - **0: Bidirectional**
  - **1: Unidirectional**
- **G47**: **Safety clearance**
  Further information: "Safety clearance G47", Page 203
- **T**: **Tool number** – turret pocket number
- **G14**: **Tool change point**
  Further information: "Tool change point G14", Page 203
- **ID**: **ID no.**
- **S**: **Cutting speed** or **Constant speed**
- **F**: **Feed per revolution**
- **MT**: **M after T**: **M** function that is executed after the **T** tool call
- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step
- **MFE**: **M at end**: **M** function that is executed at the end of the machining step
- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
  - Main drive
  - Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)

> 🛈 Type of machining for technology database access: **Recess turning**

Cycle run:

1 Calculate the proportioning of cuts
2 Approach the workpiece from the **Start point** for the first pass
3 Execute the first cut (recessing)
4 Machine perpendicularly to the recessing direction (turning)
5 Repeat 3 to 4 until the **End point contour X2**, **Z2** is reached
6 Return to the **Start point** on a paraxial path
7 Move to the **Tool change point** according to the setting in **G14**

Radial recess turning – Expanded

> ▶ Select **Recessing cycle**

> ▶ Select **Recess turning**

> ▶ Select **Radial recess turning**

> ▶ Select the **Expanded** soft key

Taking the oversizes into account, the cycle machines the area described by the **Start point X**/**Start point contour Z1**, and the **End point contour**.

Further information: "Recess turning", Page 301

Cycle parameters:

- **X**, **Z**: **Start point**
- **X1**, **Z1**: **Start point contour**
- **X2**, **Z2**: **End point contour**
- **P**: **Feed depth** – Maximum infeed depth
- **O**: **Recessing feed** (default: active feed rate)
- **I**, **K**: **O-size X** and **Z**
- **A**: **Start angle** (range: 0° <= **A** < 90°; default: 0°)
- **W**: **Final angle** (range: 0° <= **W** < 90°; default: 0°)
- **R**: **Rounding**
- **T**: **Tool number** – turret pocket number
- **G14**: **Tool change point**
  Further information: "Tool change point G14", Page 203
- **ID**: **ID no.**
- **S**: **Cutting speed** or **Constant speed**
- **F**: **Feed per revolution**
- **B1**, **B2**: **-B cham./+B round.** (**B1** at contour start and **B2** at contour end)
  - **B** > 0: Radius of rounding
  - **B** < 0: Width of chamfer
- **B**: **Offset width** (default: 0)
- **U**: **Unidir. turning** (default: 0)
  - **0: Bidirectional**
  - **1: Unidirectional**
- **G47**: **Safety clearance**
  Further information: "Safety clearance G47", Page 203
- **MT**: **M after T**: **M** function that is executed after the **T** tool call
- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step
- **MFE**: **M at end**: **M** function that is executed at the end of the machining step

- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
  - Main drive
  - Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)

> **i** Type of machining for technology database access: **Recess turning**

Cycle run:
1 Calculate the proportioning of cuts
2 Approach the workpiece from the **Start point** for the first pass
3 Execute the first cut (recessing)
4 Machine perpendicularly to the recessing direction (turning)
5 Repeat 3 to 4 until the **End point contour X2**, **Z2** is reached
6 Machine a chamfer or rounding at the contour start or contour end, if defined
7 Return to the **Start point** along a paraxial path
8 Move to the **Tool change point** according to the setting in **G14**

Axial recess turning – Expanded
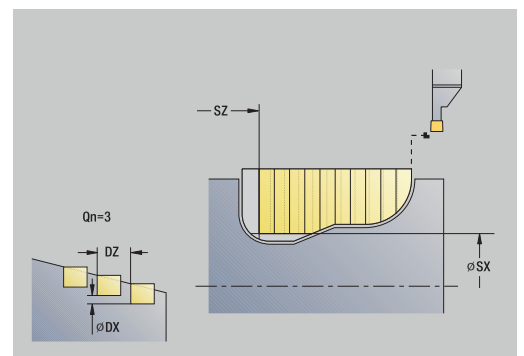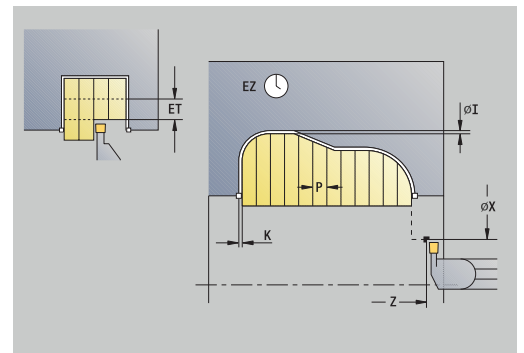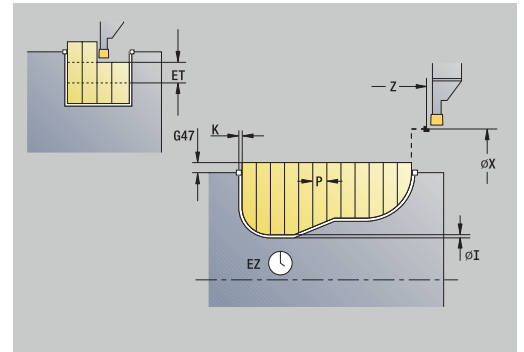
▶ Select **Recessing cycle**

▶ Select **Recess turning**

▶ Select **Axial recess turning**

▶ Select the **Expanded** soft key

Taking the oversizes into account, the cycle machines the area described by the **Start point Z/Start point contour X1**, and the **End point contour**.

Further information: "Recess turning", Page 301

Cycle parameters:

- **X**, **Z**: **Start point**
- **X1**, **Z1**: **Start point contour**
- **X2**, **Z2**: **End point contour**
- **P**: **Feed depth** – Maximum infeed depth
- **O**: **Recessing feed** (default: active feed rate)
- **I**, **K**: **O-size X** and **Z**
- **A**: **Start angle** (range: 0° <= **A** < 90°; default: 0°)
- **W**: **Final angle** (range: 0° <= **W** < 90°; default: 0°)
- **R**: **Rounding**
- **T**: **Tool number** – turret pocket number
- **G14**: **Tool change point**
  Further information: "Tool change point G14", Page 203
- **ID**: **ID no.**
- **S**: **Cutting speed** or **Constant speed**
- **F**: **Feed per revolution**
- **B1**, **B2**: **-B cham./+B round.** (**B1** at contour start and **B2** at contour end)
  - **B** > 0: Radius of rounding
  - **B** < 0: Width of chamfer
- **B**: **Offset width** (default: 0)
- **U**: **Unidir. turning** (default: 0)
  - **0: Bidirectional**
  - **1: Unidirectional**
- **G47**: **Safety clearance**
  Further information: "Safety clearance G47", Page 203
- **MT**: **M after T**: **M** function that is executed after the **T** tool call
- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step
- **MFE**: **M at end**: **M** function that is executed at the end of the machining step

- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
  - Main drive
  - Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)

> **ℹ** Type of machining for technology database access: **Recess turning**

Cycle run:

1 Calculate the proportioning of cuts
2 Approach the workpiece from the **Start point** for the first pass
3 Execute the first cut (recessing)
4 Machine perpendicularly to the recessing direction (turning)
5 Repeat 3 to 4 until the **End point contour X2**, **Z2** is reached
6 Machine a chamfer or rounding at the contour start or contour end, if defined
7 Return to the **Start point** on a paraxial path
8 Move to the **Tool change point** according to the setting in **G14**

Radial recess turning, finishing
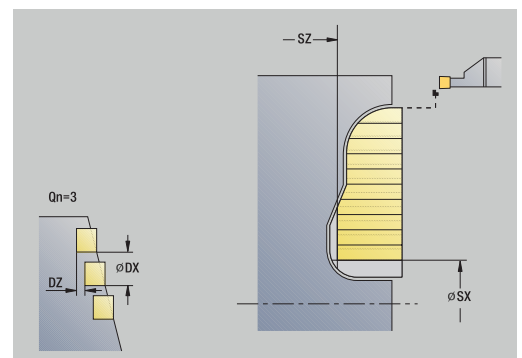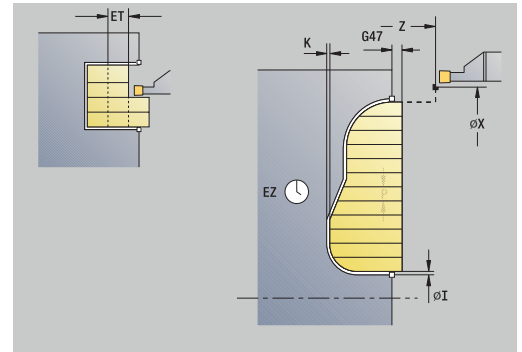
> ▶ Select **Recessing cycle**

> ▶ Select **Recess turning**

> ▶ Select **Radial recess turning**

Finishing
run

> ▶ Press the **Finishing run** soft key

The cycle finishes the contour area defined by the **Start point** and the **End point contour**.

Further information: "Recess turning", Page 301

> ℹ With **oversizes I**, **K**, you define the material left remaining after the finishing cycle.

Cycle parameters:
- **X**, **Z**: **Start point**
- **X2**, **Z2**: **End point contour**
- **I**, **K**: **Oversize blank X** and **Z**
- **G47**: **Safety clearance**
  Further information: "Safety clearance G47", Page 203
- **T**: **Tool number** – turret pocket number
- **G14**: **Tool change point**
  Further information: "Tool change point G14", Page 203
- **ID**: **ID no.**
- **S**: **Cutting speed** or **Constant speed**
- **F**: **Feed per revolution**
- **MT**: **M after T**: **M** function that is executed after the **T** tool call
- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step
- **MFE**: **M at end**: **M** function that is executed at the end of the machining step
- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
  - Main drive
  - Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)

> ℹ Type of machining for technology database access: **Recess turning**

Cycle run:

1 Approach the workpiece from the **Start point** on a paraxial path
2 Finish the first side and the contour area up to a position just before the **End point contour X2**, **Z2**
3 Move to the **Start point X**/**End point contour Z2** on a paraxial path
4 Finish the second side, then finish the remainder of the contour valley
5 Return to the **Start point** on a paraxial path
6 Move to the **Tool change point** according to the setting in **G14**

Axial recess turning, finishing

▶ Select **Recessing cycle**

▶ Select **Recess turning**

▶ Select **Axial recess turning**

| Finishing run | ▶ Press the **Finishing run** soft key |

The cycle finishes the contour area defined by the **Start point** and the **End point contour**.

Further information: "Recess turning", Page 301

> **i** With **oversizes I**, **K**, you define the material left remaining after the finishing cycle.

Cycle parameters:
- **X**, **Z**: **Start point**
- **X2**, **Z2**: **End point contour**
- **I**, **K**: **Oversize blank X** and **Z**
- **G47**: **Safety clearance**
  Further information: "Safety clearance G47", Page 203
- **T**: **Tool number** – turret pocket number
- **G14**: **Tool change point**
  Further information: "Tool change point G14", Page 203
- **ID**: **ID no.**
- **S**: **Cutting speed** or **Constant speed**
- **F**: **Feed per revolution**
- **MT**: **M after T**: **M** function that is executed after the **T** tool call
- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step
- **MFE**: **M at end**: **M** function that is executed at the end of the machining step
- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
  - Main drive
  - Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)

> **i** Type of machining for technology database access: **Recess turning**

Cycle run:

1 Approach the workpiece from the **Start point** along a paraxial path

2 Finish the first side and the contour area up to a position just before the **End point contour X2**, **Z2**

3 Move along a paraxial path to the **Start point Z**/**End point contour X2**

4 Finish the second side, then finish the remainder of the contour valley

5 Return to the **Start point** along a paraxial path

6 Move to the **Tool change point** according to the setting in **G14**

Radial recess turning, finishing—Expanded

▶ Select **Recessing cycle**

▶ Select **Recess turning**

▶ Select **Radial recess turning**

Expanded   ▶ Select the **Expanded** soft key

Finishing   ▶ Press the **Finishing run** soft key
run

The cycle finishes the contour area defined by the **Start point contour** and the **End point contour**.

Further information: "Recess turning", Page 301

> ■ With **workpiece-blank oversizes RI**, **RK**, you define the material to be machined during the finishing cycle. For this purpose, enter the oversizes for recess turning, finishing
> ■ With **oversizes I**, **K**, you define the material left remaining after the finishing cycle

Cycle parameters:
- **X**, **Z**: **Start point**
- **X1**, **Z1**: **Start point contour**
- **X2**, **Z2**: **End point contour**
- **RB**: **Depth compens.**
- **I**, **K**: **O-size X** and **Z**
- **A**: **Start angle** (range: 0° <= **A** < 90°; default: 0°)
- **W**: **Final angle** (range: 0° <= **W** < 90°; default: 0°)
- **R**: **Rounding**
- **T**: **Tool number** – turret pocket number
- **G14**: **Tool change point**
  Further information: "Tool change point G14", Page 203
- **ID**: **ID no.**
- **S**: **Cutting speed** or **Constant speed**
- **F**: **Feed per revolution**
- **B1**, **B2**: **-B cham./+B round.** (**B1** at contour start and **B2** at contour end)
  - **B** > 0: Radius of rounding
  - **B** < 0: Width of chamfer
- **RI**, **RK**: **Oversize blank X** and **Z** – Oversize before the finishing operation for calculating the approach and departure paths and the finishing area
- **G47**: **Safety clearance**
  Further information: "Safety clearance G47", Page 203
- **MT**: **M after T**: **M** function that is executed after the **T** tool call
- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step

- **MFE**: **M at end**: **M** function that is executed at the end of the machining step
- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
  - Main drive
  - Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)

> **i** Type of machining for technology database access: **Recess turning**

Cycle run:
1. Approach the workpiece from the **Start point**
2. Finish the first side, taking the optional contour elements into account; then finish the contour valley up to a position just before the **End point contour X2**, **Z2**
3. Approach the workpiece on a paraxial path for finishing the second side
4. Finish the second side, taking the optional contour elements into account; then finish the remainder of the contour valley
5. Finish the chamfer or rounding at the contour start or contour end, if defined
6. Move to the **Tool change point** according to the setting in **G14**

Axial recess turning, finishing − Expanded

▶ Select **Recessing cycle**

▶ Select **Recess turning**

▶ Select **Axial recess turning**

▶ Select the **Expanded** soft key

▶ Press the **Finishing run** soft key

The cycle finishes the contour area defined by the **Start point contour** and the **End point contour**.

Further information: "Recess turning", Page 301

> ⓘ ■ With **workpiece-blank oversizes RI**, **RK**, you define the material to be machined during the finishing cycle. For this purpose, enter the oversizes for recess turning, finishing
> ■ With **oversizes I**, **K**, you define the material left remaining after the finishing cycle

Cycle parameters:
- **X**, **Z**: **Start point**
- **X1**, **Z1**: **Start point contour**
- **X2**, **Z2**: **End point contour**
- **RB**: **Depth compens.**
- **I**, **K**: **O-size X** and **Z**
- **A**: **Start angle** (range: 0° <= **A** < 90°; default: 0°)
- **W**: **Final angle** (range: 0° <= **W** < 90°; default: 0°)
- **R**: **Rounding**
- **T**: **Tool number** – turret pocket number
- **G14**: **Tool change point**
  Further information: "Tool change point G14", Page 203
- **ID**: **ID no.**
- **S**: **Cutting speed** or **Constant speed**
- **F**: **Feed per revolution**
- **B1**, **B2**: **-B cham./+B round.** (**B1** at contour start and **B2** at contour end)
  - **B** > 0: Radius of rounding
  - **B** < 0: Width of chamfer
- **RI**, **RK**: **Oversize blank X** and **Z** – Oversize before the finishing operation for calculating the approach and departure paths and the finishing area
- **G47**: **Safety clearance**
  Further information: "Safety clearance G47", Page 203
- **MT**: **M after T**: **M** function that is executed after the **T** tool call
- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step

- **MFE**: **M at end**: **M** function that is executed at the end of the machining step
- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
    - Main drive
    - Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)

> **ℹ** Type of machining for technology database access: **Recess turning**

Cycle run:

1 Approach the workpiece from the **Start point**
2 Finish the first side, taking the optional contour elements into account; then finish the contour valley up to a position just before the **End point contour X2**, **Z2**
3 Approach the workpiece along a paraxial path for finishing the second side
4 Finish the second side, taking the optional contour elements into account; then finish the remainder of the contour valley
5 Finish the chamfer or rounding at the contour start or contour end, if defined
6 Move to the **Tool change point** according to the setting in **G14**

ICP-Recess turning long.

▶ Select **Recessing cycle**

▶ Select **Recess turning**

▶ Select **ICP-Recess turning long.**

The cycle machines the defined area.
Further information: "Recess turning", Page 301

> ⓘ ▪ If you are machining **descending contours,** define
> the **Start point**—not the **Start point blank**. The cycle
> machines the area defined by the **Start point** and the
> ICP contour, taking the oversizes into account
> ▪ If you are machining **ascending contours,** define the
> **Start point** and the **Start point blank**. The cycle
> machines the area defined by the **Start point blank** and
> the ICP contour, taking the oversizes into account

Cycle parameters:

▪ **X**, **Z**: **Start point**
▪ **X1**, **Z1**: **Start point blank**
▪ **FK**: **No. of ICP finished part** – name of the contour to be
  machined
▪ **P**: **Feed depth** – Maximum infeed depth
▪ **O**: **Recessing feed** (default: active feed rate)
▪ **I**, **K**: **O-size X** and **Z**
▪ **SX**, **SZ**: **Cutting limit in X** and **Z**
  Further information: "Cutting limits SX, SZ", Page 203
▪ **B**: **Offset width** (default: 0)
▪ **U**: **Unidir. turning** (default: 0)
  ▪ **0: Bidirectional**
  ▪ **1: Unidirectional**
▪ **T**: **Tool number** – turret pocket number
▪ **G14**: **Tool change point**
  Further information: "Tool change point G14", Page 203
▪ **ID**: **ID no.**
▪ **S**: **Cutting speed** or **Constant speed**
▪ **F**: **Feed per revolution**
▪ **A**: **Start angle** (default: opposite to recessing direction)
▪ **W**: **Depart.angle** (default: opposite to recessing direction)
▪ **G47**: **Safety clearance**
  Further information: "Safety clearance G47", Page 203
▪ **MT**: **M after T**: **M** function that is executed after the **T** tool call
▪ **MFS**: **M at beginning**: **M** function that is executed at the beginning
  of the machining step
▪ **MFE**: **M at end**: **M** function that is executed at the end of the
  machining step

- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
  - Main drive
  - Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)

> ℹ Type of machining for technology database access: **Recess turning**

Cycle run:

1. Calculate the proportioning of cuts
2. Approach the workpiece from the **Start point** for the first pass
3. Execute the first cut (recessing)
4. Machine perpendicularly to the recessing direction (turning)
5. Repeat 3 to 4 until the defined area has been machined
6. Return to the **Start point** along a paraxial path
7. Move to the **Tool change point** according to the setting in **G14**

ICP-Recess turning transv.

▶ Select **Recessing cycle**
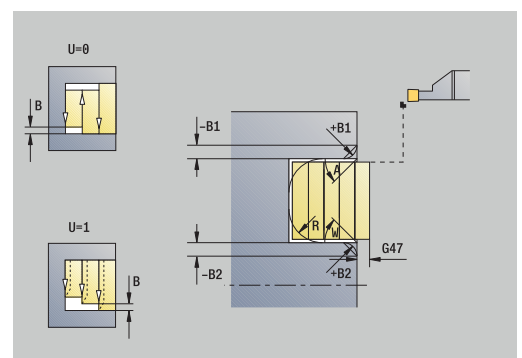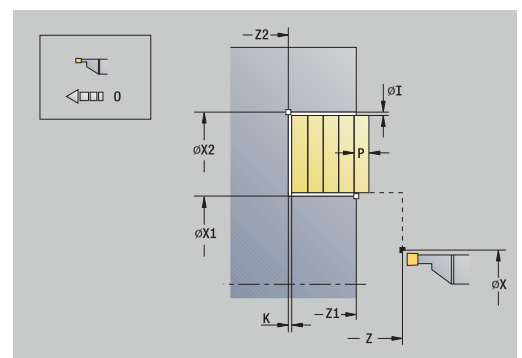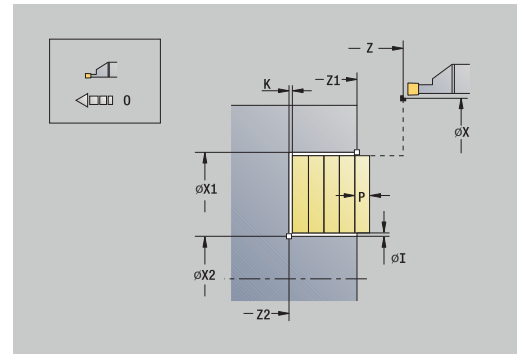
▶ Select **Recess turning**

▶ Select **ICP-Recess turning transv.**

The cycle machines the defined area.

Further information: "Recess turning", Page 301

> ⓘ ▪ If you are machining **descending contours,** define the **Start point**—not the **Start point blank**. The cycle machines the area defined by the **Start point** and the ICP contour, taking the oversizes into account
> ▪ If you are machining **ascending contours,** define the **Start point** and the **Start point blank**. The cycle machines the area defined by the **Start point blank** and the ICP contour, taking the oversizes into account

Cycle parameters:

- **X**, **Z**: **Start point**
- **X1**, **Z1**: **Start point blank**
- **FK**: **No. of ICP finished part** – name of the contour to be machined
- **P**: **Feed depth** – Maximum infeed depth
- **O**: **Recessing feed** (default: active feed rate)
- **I**, **K**: **O-size X** and **Z**
- **SX**, **SZ**: **Cutting limit in X** and **Z**
  Further information: "Cutting limits SX, SZ", Page 203
- **B**: **Offset width** (default: 0)
- **U**: **Unidir. turning** (default: 0)
  - **0: Bidirectional**
  - **1: Unidirectional**
- **T**: **Tool number** – turret pocket number
- **G14**: **Tool change point**
  Further information: "Tool change point G14", Page 203
- **ID**: **ID no.**
- **S**: **Cutting speed** or **Constant speed**
- **F**: **Feed per revolution**
- **A**: **Start angle** (default: opposite to recessing direction)
- **W**: **Depart.angle** (default: opposite to recessing direction)
- **G47**: **Safety clearance**
  Further information: "Safety clearance G47", Page 203
- **MT**: **M after T**: **M** function that is executed after the **T** tool call
- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step
- **MFE**: **M at end**: **M** function that is executed at the end of the machining step

- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
    - Main drive
    - Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)

> Type of machining for technology database access: **Recess turning**

Cycle run:

1 Calculate the proportioning of cuts
2 Approach the workpiece from the **Start point** for the first pass
3 Execute the first cut (recessing)
4 Machine perpendicularly to the recessing direction (turning)
5 Repeat 3 to 4 until the defined area has been machined
6 Return to the **Start point** on a paraxial path
7 Move to the **Tool change point** according to the setting in **G14**

ICP recess turning, radial finishing

▶   Select **Recessing cycle**

▶   Select **Recess turning**

▶   Select **ICP-Recess turning long.**

Finishing run   ▶   Press the **Finishing run** soft key

The cycle finishes the contour area defined by the ICP contour. At the end of the cycle, the tool returns to the **Start point**.
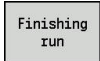
Further information: "Recess turning", Page 301

ℹ️   ■   With **workpiece-blank oversizes RI**, **RK**, you define the material to be machined during the finishing cycle. For this purpose, enter the oversizes for recess turning, finishing
    ■   With **oversizes I**, **K**, you define the material left remaining after the finishing cycle

Cycle parameters:
- **X**, **Z**: **Start point**
- **FK**: **No. of ICP finished part** – name of the contour to be machined
- **RB**: **Depth compens.**
- **I**, **K**: **O-size X** and **Z**
- **SX**, **SZ**: **Cutting limit in X** and **Z**
  Further information: "Cutting limits SX, SZ", Page 203
- **RI**, **RK**: **Oversize blank X** and **Z**
- **T**: **Tool number** – turret pocket number
- **G14**: **Tool change point**
  Further information: "Tool change point G14", Page 203
- **ID**: **ID no.**
- **S**: **Cutting speed** or **Constant speed**
- **F**: **Feed per revolution**
- **A**: **Start angle** (default: opposite to recessing direction)
- **W**: **Depart.angle** (default: opposite to recessing direction)
- **G47**: **Safety clearance**
  Further information: "Safety clearance G47", Page 203
- **MT**: **M after T**: **M** function that is executed after the **T** tool call
- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step
- **MFE**: **M at end**: **M** function that is executed at the end of the machining step

- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
  - Main drive
  - Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)

> **ℹ** Type of machining for technology database access: **Recess turning**

Cycle run:

1. Pre-position from the **Start point** to the **Safety clearance** above the first side along a paraxial path
2. Finish the complete contour in one cut
3. Return to the **Start point** along a paraxial path
4. Move to the **Tool change point** according to the setting in **G14**

ICP recess turning, axial finishing

▶ Select **Recessing cycle**

▶ Select **Recess turning**

▶ Select **ICP-Recess turning transv.**

Finishing run   ▶ Press the **Finishing run** soft key

The cycle finishes the contour area defined by the ICP contour. At the end of the cycle, the tool returns to the **Start point**.

Further information: "Recess turning", Page 301

> ℹ
> ■ With **workpiece-blank oversizes RI**, **RK**, you define the material to be machined during the finishing cycle. For this purpose, enter the oversizes for recess turning, finishing
> ■ With **oversizes I**, **K**, you define the material left remaining after the finishing cycle

Cycle parameters:

- **X**, **Z**: **Start point**
- **FK**: **No. of ICP finished part** – name of the contour to be machined
- **RB**: **Depth compens.**
- **I**, **K**: **O-size X** and **Z**
- **SX**, **SZ**: **Cutting limit in X** and **Z**
  Further information: "Cutting limits SX, SZ", Page 203
- **RI**, **RK**: **Oversize blank X** and **Z**
- **T**: **Tool number** – turret pocket number
- **G14**: **Tool change point**
  Further information: "Tool change point G14", Page 203
- **ID**: **ID no.**
- **S**: **Cutting speed** or **Constant speed**
- **F**: **Feed per revolution**
- **A**: **Start angle** (default: opposite to recessing direction)
- **W**: **Depart.angle** (default: opposite to recessing direction)
- **G47**: **Safety clearance**
  Further information: "Safety clearance G47", Page 203
- **MT**: **M after T**: **M** function that is executed after the **T** tool call
- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step
- **MFE**: **M at end**: **M** function that is executed at the end of the machining step

- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
  - Main drive
  - Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)

> **i** Type of machining for technology database access: **Recess turning**

Cycle run:

1. Pre-position from the **Start point** to the **Safety clearance** above the first side along a paraxial path
2. Finish the complete contour in one cut
3. Return to the **Start point** along a paraxial path
4. Move to the **Tool change point** according to the setting in **G14**

Undercut. Form H

|icon| ▶ Select **Recessing cycle**

|icon| ▶ Select **Undercut H**

The contour depends on the parameters defined. If you do not define the **Undercut radius**, the oblique cut will be executed up to **Contour corner Z1** (tool radius = **Undercut radius**).

If you do not define the **plunging angle,** it is calculated from the **Undercut lgth.** and **Undercut radius**. The final point of the undercut is then located at the **Contour corner**.

The end point of the undercut is determined from the plunging angle in accordance with **Undercut type H.**

Cycle parameters:

- **X**, **Z**: **Start point**
- **X1**, **Z1**: **Contour corner**
- **K**: **Undercut lgth.**
- **R**: **Undercut radius** (default: no circular element)
- **W**: **Plunging angle** (default: **W** is calculated)
- **G47**: **Safety clearance**
  Further information: "Safety clearance G47", Page 203
- **T**: **Tool number** – turret pocket number
- **G14**: **Tool change point**
  Further information: "Tool change point G14", Page 203
- **ID**: **ID no.**
- **S**: **Cutting speed** or **Constant speed**
- **F**: **Feed per revolution**
- **MT**: **M after T**: **M** function that is executed after the **T** tool call
- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step
- **MFE**: **M at end**: **M** function that is executed at the end of the machining step
- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
  - Main drive
  - Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)

> **i** Type of machining for technology database access:
> **Finishing**

Cycle run:

1 Approach the workpiece from the **Start point** up to the safety clearance
2 Machine the undercut according to the cycle parameters
3 Return to the **Start point** on a diagonal path
4 Move to the **Tool change point** according to the setting in **G14**

Undercut. Form K

▶ Select **Recessing cycle**

▶ Select **Undercut K**

This cycle performs only one linear cut at an angle of 45°. The resulting contour geometry therefore depends on the tool that is used.

Cycle parameters:

- **X**, **Z**: **Start point**
- **X1**, **Z1**: **Contour corner**
- **I**: **Undercut depth**
- **G47**: **Safety clearance**
  Further information: "Safety clearance G47", Page 203
- **T**: **Tool number** – turret pocket number
- **G14**: **Tool change point**
  Further information: "Tool change point G14", Page 203
- **ID**: **ID no.**
- **S**: **Cutting speed** or **Constant speed**
- **F**: **Feed per revolution**
- **MT**: **M after T**: **M** function that is executed after the **T** tool call
- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step
- **MFE**: **M at end**: **M** function that is executed at the end of the machining step
- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
  - Main drive
  - Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)

> ⓘ   Type of machining for technology database access:
> **Finishing**

Cycle run:

1. Pre-position at an angle of 45° to the **Safety clearance** above the **Contour corner X1**, **Z1** at rapid traverse
2. Plunge by **Undercut depth I**
3. Return the tool to the **Start point** on the same path
4. Move to the **Tool change point** according to the setting in **G14**

Undercut. Form U

▶ Select **Recessing cycle**

▶ Select **Undercut U**

This cycle machines an **Undercut type U** and, if programmed, finishes the adjoining plane surface. The undercut is machined in several passes if the **Undercut width** is greater than the cutting width of the tool. If the cutting width of the tool is not defined, the control assumes that the tool's cutting width equals the **Undercut width**. Either a chamfer or a rounding arc can be machined.

Cycle parameters:

- **X**, **Z**: **Start point**
- **X1**, **Z1**: **Contour corner**
- **X2**: **End point transv.**
- **I**: **Undercut diameter**
- **K**: **Undercut width**
- **B**: **-B cham./+B round.**
    - **B** > 0: Radius of rounding
    - **B** < 0: Width of chamfer
- **G47**: **Safety clearance**
  Further information: "Safety clearance G47", Page 203
- **T**: **Tool number** – turret pocket number
- **G14**: **Tool change point**
  Further information: "Tool change point G14", Page 203
- **ID**: **ID no.**
- **S**: **Cutting speed** or **Constant speed**
- **F**: **Feed per revolution**
- **MT**: **M after T**: **M** function that is executed after the **T** tool call
- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step
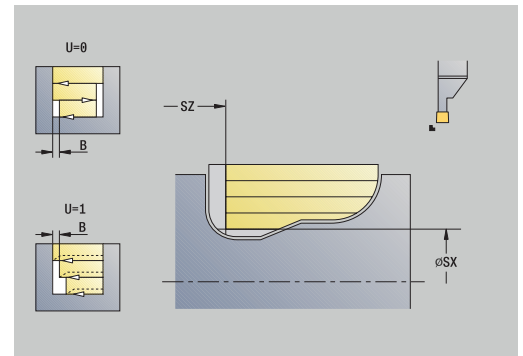- **MFE**: **M at end**: **M** function that is executed at the end of the machining step
- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
    - Main drive
    - Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)

> **ⓘ** Type of machining for technology database access:
> **Finishing**

Cycle run:

1 Calculate the proportioning of cuts
2 Approach the workpiece from the **Start point** up to the safety clearance
3 Move at the programmed feed rate to the **Undercut diameter I** and dwell at this position (for two revolutions)
4 Retract and approach for the next pass
5 Repeat 3 to 4 until the **Contour corner Z1** is reached
6 After the last pass, finish the adjoining plane surface, starting from **End point transv. X2**, if defined
7 Machine the chamfer or rounding arc, if defined
8 Return to the **Start point** on a diagonal path
9 Move to the **Tool change point** according to the setting in **G14**

Cut off

▶ Select **Recessing cycle**

▶ Select **Cut off**

The cycle parts the workpiece. If programmed, a chamfer or rounding arc is machined on the outside diameter.

Cycle parameters:

- **X**, **Z**: **Start point**
- **X1**, **Z1**: **Contour corner**
- **XE**: **Inner diameter (pipe)**
- **D**: **Maximum speed**
- **I**: **Diam. feed reduction** – Limit diameter above which the workpiece will be machined with a reduced feed rate
- **B**: **-B cham./+B round.**
  - **B** > 0: Radius of rounding
  - **B** < 0: Width of chamfer
- **E**: **Reduced feed**
- **K**: **Retraction distance** after parting – Lift off the tool laterally from the plane surface before retraction
- **SD**: **Speed limit from I up**
- **U**: **Collector active diameter** (machine-dependent)
- **G47**: **Safety clearance**
  Further information: "Safety clearance G47", Page 203
- **T**: **Tool number** – turret pocket number
- **G14**: **Tool change point**
  Further information: "Tool change point G14", Page 203
- **ID**: **ID no.**
- **S**: **Cutting speed** or **Constant speed**
- **F**: **Feed per revolution**
- **MT**: **M after T**: **M** function that is executed after the **T** tool call
- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step
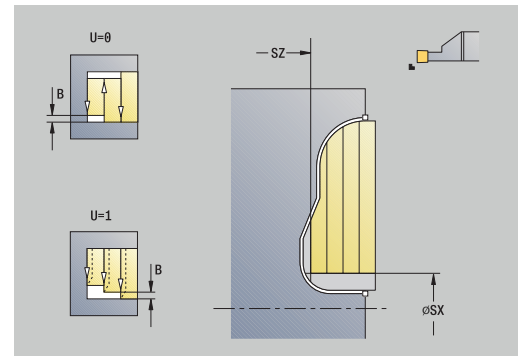- **MFE**: **M at end**: **M** function that is executed at the end of the machining step

- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
  - Main drive
  - Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)

> **ⓘ** Type of machining for technology database access: **Cut off**

Cycle run:

1 Approach the workpiece from the **Start point** up to the **Safety clearance**
2 Cut to the depth of the chamfer or rounding and machine the chamfer/rounding if defined
3 Depending on the cycle parameters, move at the programmed feed rate to
  - The turning center, or
  - The **Inner diameter (pipe) XE**

    If you have programmed a feed-rate reduction, the control switches to the **Reduced feed E** as soon as the tool reaches the **Diam. feed reduction I**.
4 Retract along the plane surface and return to the **Start point**
5 Move to the **Tool change point** according to the setting in **G14**

> **ⓘ** The limit to the **Maximum speed D** is only effective in the cycle. After the cycle ends, the speed limit that was valid before the cycle becomes effective again.

Examples of recessing cycles

Recess outside

The machining operation will be executed with the **Recessing radial E** cycle, taking oversizes into account. This contour area will be finished subsequently with **Recessing radial finishing E**.

The rounding arcs in the corners of the contour valley and the oblique surfaces at the contour start and end are also machined in **expanded mode.**

Be sure to enter the correct values for the **Start point contour X1**, **Z1** and **End point contour X2**, **Z2** parameters. These parameters determine the cutting and infeed directions—in this example, outside machining and infeed in negative Z-axis direction.

Tool data:

- Turning tool (for external machining)
- **TO** = 1 – Tool orientation
- **SB** = 4 – Cutting width (4 mm)
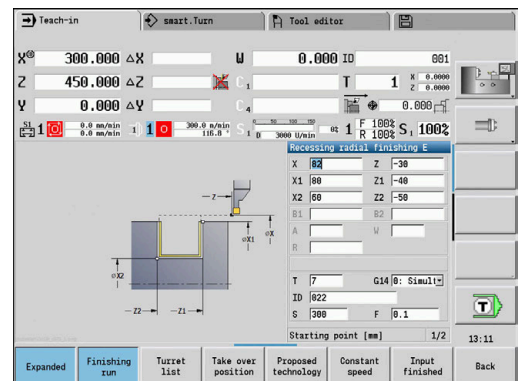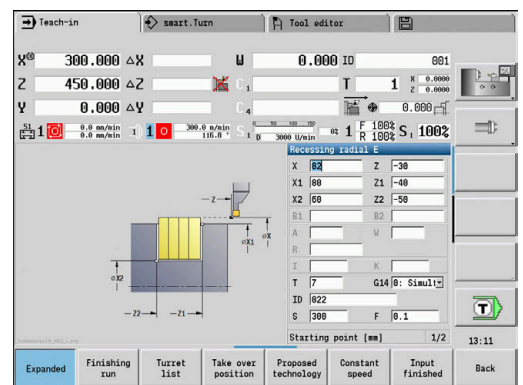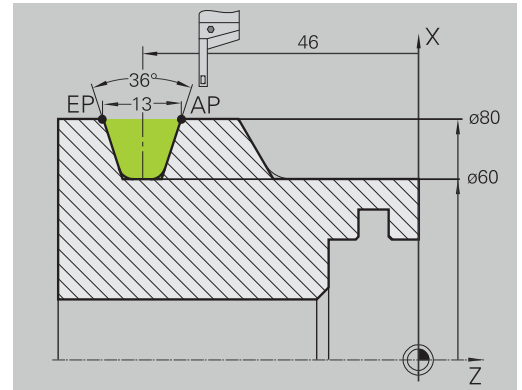
Recess inside

The machining operation will be executed with the **Recessing radial E** cycle, taking oversizes into account. This contour area will be finished subsequently with **Recessing radial finishing E**.

Since the **Cut. width P** is not entered, the control plunge-cuts with 80 % of the cutting width of the tool.

In **expanded mode,** the chamfers are machined at the start/end of the contour.

Be sure to enter the correct values for the **Start point contour X1**, **Z1**, and **End point contour X2**, **Z2** parameters. These parameters determine the cutting and infeed directions—in this example, inside machining and infeed in negative Z-axis direction.

Tool data:

- Turning tool (for internal machining)
- **TO** = 7 – Tool orientation
- **SB** = 2 – Cutting width (2 mm)
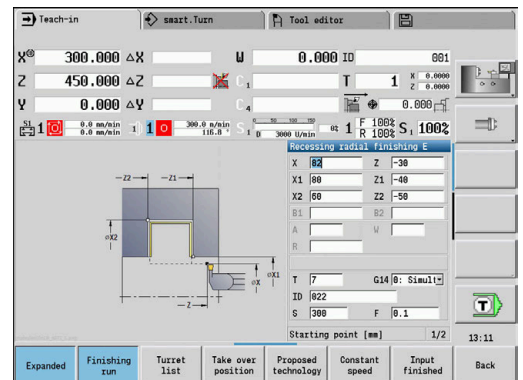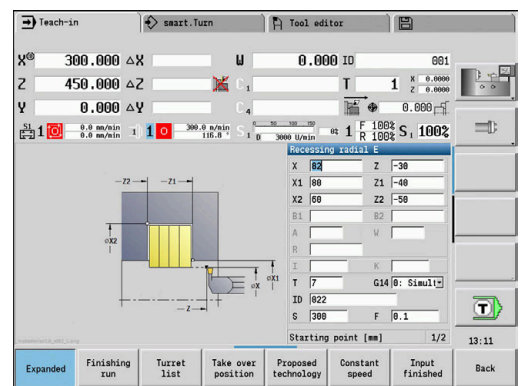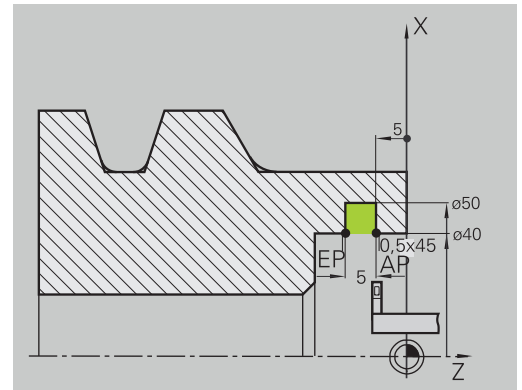
7.6 Thread and undercut cycles

Menu item	Meaning
	The thread and undercut cycles machine single or multi-start longitudinal and tapered threads, as well as thread undercuts

In Teach-in mode you can

■ Repeat the **last cut** to compensate for tool inaccuracies
■ Use the **Thread Recutting** software option (option 11) to rework damaged threads (only in **Machine** operating mode)

ℹ️
■ Threads are cut with constant rotational speed
■ Feed rate override is not effective during cycle execution

⚙️
Refer to your machine manual.

The machine manufacturer specifies whether the tool is retracted immediately after **NC stop** or the thread is completed.
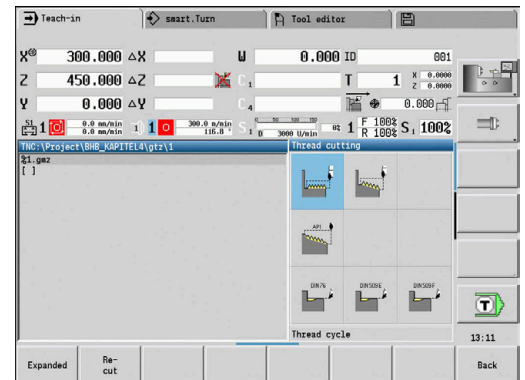
Menu item	Thread and undercut cycles
	Thread. cycle Longitudinal single or multi-start thread
	Tapered thr. Tapered single or multi-start thread
	API thread Single or multi-start API thread (API: American Petroleum Institute)
	Undercut DIN 76 Thread undercut and thread chamfer
	Undercut DIN 509E Undercut and cylinder chamfer
	Undercut DIN 509F Undercut and cylinder chamfer

Direction of threads

The control determines the direction of the thread from the parameters for **Starting point Z** (in **Machine** operating mode: current tool position) and **End point thread Z2**. You select internal or external thread by soft key.

GV parameter: Type of infeed

With the **GV** parameter, you define the type of infeed for thread cutting cycles.

The following infeed types are available:

- **0: Const. mach. X-section** – The control reduces the cutting depth with each infeed so that the chip cross section, and therefore the removal rate, remain constant

- **1: Const. infeed** – With each infeed the control uses the same cutting depth without exceeding the **Max. approach I**

- **2: EPL with distrib. of cuts** – The control calculates the cutting depth for constant infeed from **Thread pitch F1** and **Shaft speed constant S**. If the multiple of the cutting depth does not correspond to the **Thread depth**, the control uses the **Rem. cut depth** for the first infeed. The control proportions the remaining cuts, i.e. it divides the last cutting depth into four partial cuts. The first cut is half the calculated cutting depth, the second is a quarter and the third and fourth each are an eighth

- **3: EPL w/o distrib. of cuts** – The control calculates the cutting depth for constant infeed from **Thread pitch F1** and **Shaft speed constant S**. If the multiple of the cutting depth does not correspond to the **Thread depth**, the control uses the **Rem. cut depth** for the first infeed. All subsequent infeeds are constant and correspond to the calculated cutting depth

- **4: MANUALplus 4110** – The control executes the first infeed with the **Max. approach I**. To determine the subsequent cutting depths, the control uses the formula **gt** = 2 * **I** * SQRT "current no. of cuts", where **gt** is the absolute depth. Since the cutting depth is reduced with each infeed because the current number of cuts is increased accordingly by the value 1, and if the cutting depth falls below the value given in **Rem. cut depth R**, the control uses the value defined here as the new constant cutting depth. If the multiple of the cutting depth does not correspond to the **Thread depth**, the control executes the last cut at the final depth

- **5: Constant infeed (4290)** – With each infeed, the control uses the same cutting depth where the cutting depth corresponds to **Max. approach I**. If the multiple of the cutting depth does not correspond to the **Thread depth**, the control uses the **Rem. cut depth** for the first infeed

- **6: Const. w/ distrib. (4290)** – With each infeed, the control uses the same cutting depth where the cutting depth corresponds to **Max. approach I**. If the multiple of the cutting depth does not correspond to the **Thread depth**, the control uses the **Rem. cut depth** for the first infeed. The control proportions the remaining cuts, i.e. it divides the last cutting depth into four partial cuts. The first cut is half the calculated cutting depth, the second is a quarter, and the third and fourth each are an eighth

Undercut position

The control determines the position of an undercut from the parameters for **Starting point X**, **Z** (in **Machine** operating mode: current tool position), and **Start point cylinder X1**/**End point transv. Z2**.

> **ℹ** An undercut can only be machined in a right-angled paraxial contour corner in the linear axis.

Handwheel superimpositioning (option 11)

If your machine features handwheel superimposition, you can overlap axis movements during thread cutting in a limited area:

- **X direction**: Maximum programmed thread depth depending on the current cutting depth
- **Z direction**: +/- a fourth of the thread pitch

> **⚙** Refer to your machine manual.
> This function must be set up by your machine manufacturer.

> **ℹ** Remember that position changes resulting from handwheel superimposition are no longer effective after the cycle end or the **Last cut** function!

Angle of infeed, thread depth, proportioning of cuts

With some thread cycles, you can indicate the angle of infeed (thread angle). The figures show the operating sequence at an angle of infeed of −30° and an angle of infeed of 0°.

The thread depth is programmed for all thread cycles. The control reduces the cutting depth with each cut.
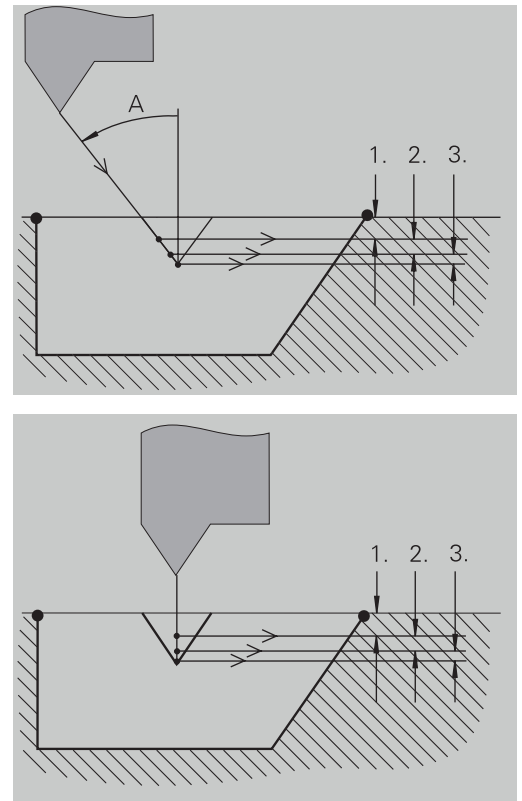
Thread run-in and thread run-out

The slide requires a run-in distance to accelerate to the programmed feed rate before starting the actual thread, and a run-out distance at the end of the thread to decelerate again.

If the run-in length or run-out length is too short, the thread may not attain the expected quality. In this case, the control displays a warning.

Last cut

After the cycle is finished, the control presents the **Last cut** option. In this way, you can enter a tool compensation value and repeat the last thread cut.

Sequence of the **Last cut** function:

Initial situation: The thread cut cycle has been run, and the thread depth is not correct.

▶ Perform the tool compensation

Last cut	▶ Press the **Last cut** soft key

	▶ Press the **NC start** key
	▶ Check the thread

> **i** The tool compensation and the **Last cut** can be repeated as often as necessary until the thread is correct.

Thread cycle (longitudinal)

▶ Select **Thread cutting**

▶ Select **Thread cycle**
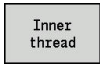
▶ Select the type of thread:
- **On:** Internal thread
- **Off:** External thread

This cycle cuts a single external or internal thread with a thread angle of 30°. Tool infeed is performed in the X axis only.

Cycle parameters:

- **X**, **Z**: **Start point** of thread
- **Z2**: **End point thread**
- **F1**: **Thread pitch** (= feed rate)
- **U**: **Thread depth** (default: no input)
  - Outside thread: **U** = 0.6134 * **F1**
  - Inside thread: **U** = −0.5413 * **F1**
- **I**: **Max. approach**
  - **I** < **U**: First cut with **I**, all subsequent cuts with cutting-depth reduction
  - **I** = **U**: One cut
  - No input: **I** is calculated from **U** and **F1**
- **G47**: **Safety clearance**
  Further information: "Safety clearance G47", Page 203
- **T**: **Tool number** – turret pocket number
- **G14**: **Tool change point**
  Further information: "Tool change point G14", Page 203
- **ID**: **ID no.**
- **S**: **Constant spindle speed**
- **GV**: **Type of infeed**
  Further information: "GV parameter: Type of infeed", Page 335
  - **0: Const. mach. X-section**
  - **1: Const. infeed**
  - **2: EPL with distrib. of cuts**
  - **3: EPL w/o distrib. of cuts**
  - **4: MANUALplus 4110**
  - **5: Constant infeed (4290)**
  - **6: Const. w/ distrib. (4290)**
- **GH**: **Type of offset**
  - **0: Without offset**
  - **1: From left**
  - **2: From right**
  - **3: Alternating left/right**

- **A**: **Approach ang.** (range: −60° < **A** < 60°; default: 30°)
  - **A** < 0: Infeed on left thread flank
  - **A** > 0: Infeed on right thread flank
- **R**: **Rem. cut depth** (only with **GV** = 4; default: 1/100 mm)
- **IC**: **Number of cuts** − The infeed is calculated from **IC** and **U**
  Usable with:
  - **GV** = 0: Constant chip cross section
  - **GV** = 1: Constant infeed
- **MT**: **M after T**: **M** function that is executed after the **T** tool call
- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step
- **MFE**: **M at end**: **M** function that is executed at the end of the machining step
- **WP**: **No. of spindle** − Displays which workpiece spindle is used to execute the cycle (machine-dependent)
  - Main drive
  - Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)

> ℹ️ Type of machining for technology database access:
> **Thread cutting**

Cycle run:
1 Calculate the proportioning of cuts
2 Start the first pass at the **Start point Z**
3 Move at programmed feed rate to **End point thread Z2**
4 Return along a paraxial path and approach for the next pass
5 Repeat 3 to 4 until the **Thread depth U** is reached
6 Move to the **Tool change point** according to the setting in **G14**

Thread cycle (longitudinal) – Expanded

▶ Select **Thread cutting**

▶ Select **Thread cycle**

▶ Select the **Expanded** soft key
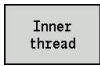
▶ Select the type of thread:
- **On:** Internal thread
- **Off:** External thread

This cycle cuts a single or multi-start external or internal thread. The thread starts at the **Start point** and ends at the **End point thread** (without a thread run-in or run-out).

Cycle parameters:

- **X**, **Z**: **Start point** of thread
- **Z2**: **End point thread**
- **F1**: **Thread pitch** (= feed rate)
- **D**: **No.gears** (default: 1 single-start thread)
- **U**: **Thread depth** (default: no input)
  - Outside thread: **U** = 0.6134 * **F1**
  - Inside thread: **U** = −0.5413 * **F1**
- **I**: **Max. approach**
  - **I** < **U**: First cut with **I**, all subsequent cuts with cutting-depth reduction
  - **I** = **U**: One cut
  - No input: **I** is calculated from **U** and **F1**
- **GK**: **Thread runout length**
- **G47**: **Safety clearance**
  Further information: "Safety clearance G47", Page 203
- **T**: **Tool number** – turret pocket number
- **G14**: **Tool change point**
  Further information: "Tool change point G14", Page 203
- **ID**: **ID no.**
- **S**: **Constant spindle speed**
- **GV**: **Type of infeed**
  Further information: "GV parameter: Type of infeed", Page 335
  - **0: Const. mach. X-section**
  - **1: Const. infeed**
  - **2: EPL with distrib. of cuts**
  - **3: EPL w/o distrib. of cuts**
  - **4: MANUALplus 4110**
  - **5: Constant infeed (4290)**
  - **6: Const. w/ distrib. (4290)**

- **GH**: **Type of offset**
    - **0: Without offset**
    - **1: From left**
    - **2: From right**
    - **3: Alternating left/right**
- **A**: **Approach ang.** (range: −60° < **A** < 60°; default: 30°)
    - **A** < 0: Infeed on left thread flank
    - **A** > 0: Infeed on right thread flank
- **R**: **Rem. cut depth** (only with **GV** = 4; default: 1/100 mm)
- **E**: **Incremental gradient** – Variable thread pitch (e.g., for manufacturing spiral conveyors or extruder shafts)
- **Q**: **Number no-load.**
- **IC**: **Number of cuts** – The infeed is calculated from **IC** and **U**
  Usable with:
    - **GV** = 0: Constant chip cross section
    - **GV** = 1: Constant infeed
- **MT**: **M after T**: **M** function that is executed after the **T** tool call
- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step
- **MFE**: **M at end**: **M** function that is executed at the end of the machining step
- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
    - Main drive
    - Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)

> **ℹ** Type of machining for technology database access:
> **Thread cutting**

Cycle run:
1 Calculate the proportioning of cuts
2 Start the first thread groove at the **Start point Z**
3 Move at programmed feed rate to **End point thread Z2**
4 Return along a paraxial path and approach for the next thread groove
5 Repeat 3 to 4 for all thread grooves
6 Approach for the next pass, taking the **reduced cutting depth** and the **Approach ang. A** into account
7 Repeat 3 to 6 until **No.gears D** and **Thread depth U** are reached
8 Move to the **Tool change point** according to the setting in **G14**

Taper thread

▶ Select **Thread cutting**
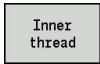
▶ Select **Taper thread**

▶ Select the type of thread:
  - **On:** Internal thread
  - **Off:** External thread

This cycle cuts a single or multi-start tapered external or internal thread.

Cycle parameters:
- **X**, **Z**: **Start point**
- **X1**, **Z1**: **Start point thread**
- **X2**, **Z2**: **End point thread**
- **F1**: **Thread pitch** (= feed rate)
- **D**: **No.gears** (default: 1 single-start thread)
- **U**: **Thread depth** (default: no input)
  - Outside thread: **U** = 0.6134 * **F1**
  - Inside thread: **U** = −0.5413 * **F1**
- **I**: **Max. approach**
  - **I** < **U**: First cut with **I**, all subsequent cuts with cutting-depth reduction
  - **I** = **U**: One cut
  - No input: **I** is calculated from **U** and **F1**
- **W**: **Taper angle** (range: −60° < **W** < 60°)
- **GK**: **Thread runout length**
  - **GK** < 0: Run-out at beginning of thread
  - **GK** > 0: Run-out at end of thread
- **G47**: **Safety clearance**
  Further information: "Safety clearance G47", Page 203
- **T**: **Tool number** – turret pocket number
- **G14**: **Tool change point**
  Further information: "Tool change point G14", Page 203
- **ID**: **ID no.**
- **S**: **Constant spindle speed**
- **GV**: **Type of infeed**
  Further information: "GV parameter: Type of infeed", Page 335
  - **0: Const. mach. X-section**
  - **1: Const. infeed**
  - **2: EPL with distrib. of cuts**
  - **3: EPL w/o distrib. of cuts**
  - **4: MANUALplus 4110**
  - **5: Constant infeed (4290)**
  - **6: Const. w/ distrib. (4290)**

- **GH**: **Type of offset**
  - **0: Without offset**
  - **1: From left**
  - **2: From right**
  - **3: Alternating left/right**
- **A**: **Approach ang.** (range: −60° < **A** < 60°; default: 30°)
  - **A** < 0: Infeed on left thread flank
  - **A** > 0: Infeed on right thread flank
- **R**: **Rem. cut depth** (only with **GV** = 4; default: 1/100 mm)
- **E**: **Incremental gradient** – Variable thread pitch (e.g., for manufacturing spiral conveyors or extruder shafts)
- **Q**: **Number no-load.**
- **IC**: **Number of cuts** – The infeed is calculated from **IC** and **U**
  Usable with:
  - **GV** = 0: Constant chip cross section
  - **GV** = 1: Constant infeed
- **MT**: **M after T**: **M** function that is executed after the **T** tool call
- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step
- **MFE**: **M at end**: **M** function that is executed at the end of the machining step
- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
  - Main drive
  - Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)

> **i** Type of machining for technology database access:
> **Thread cutting**

Parameter combinations for the **Taper angle**:
- **X1**/**Z1**, **X2**/**Z2**
- **X1**/**Z1**, **Z2**, **W**
- **Z1**, **X2**/**Z2**, **W**

Cycle run:

1 Calculate the proportioning of cuts
2 Move to the **Start point X1**, **Z1**
3 Move at programmed feed rate up to **End point thread Z2**
4 Return along a paraxial path and approach for the next thread groove
5 Repeat 3 to 4 for all thread grooves
6 Approach for the next pass, taking the **reduced cutting depth** and the **Approach ang. A** into account
7 Repeat 3 to 6 until **No.gears D** and **Thread depth U** are reached
8 Move to the **Tool change point** according to the setting in **G14**

API thread

▶ Select **Thread cutting**
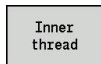
▶ Select **API thread**

▶ Select the type of thread:
  - **On:** Internal thread
  - **Off:** External thread

This cycle cuts a single or multi-start external or internal API thread. The depth of thread decreases at the thread run-out.

Cycle parameters:

- **X**, **Z**: **Start point**
- **X1**, **Z1**: **Start point thread**
- **X2**, **Z2**: **End point thread**
- **F1**: **Thread pitch** (= feed rate)
- **D**: **No.gears** (default: 1 single-start thread)
- **U**: **Thread depth** (default: no input)
  - Outside thread: **U** = 0.6134 * **F1**
  - Inside thread: **U** = −0.5413 * **F1**
- **I**: **Max. approach**
  - **I** < **U**: First cut with **I**, all subsequent cuts with cutting-depth reduction
  - **I** = **U**: One cut
  - No input: **I** is calculated from **U** and **F1**
- **WE**: **Run-out angle** (range: 0° < **WE** < 90°)
- **W**: **Taper angle** (range: −60° < **W** < 60°)
- **G47**: **Safety clearance**
  Further information: "Safety clearance G47", Page 203
- **T**: **Tool number** – turret pocket number
- **G14**: **Tool change point**
  Further information: "Tool change point G14", Page 203
- **ID**: **ID no.**
- **S**: **Constant spindle speed**
- **GV**: **Type of infeed**
  Further information: "GV parameter: Type of infeed", Page 335
  - **0: Const. mach. X-section**
  - **1: Const. infeed**
  - **2: EPL with distrib. of cuts**
  - **3: EPL w/o distrib. of cuts**
  - **4: MANUALplus 4110**
  - **5: Constant infeed (4290)**
  - **6: Const. w/ distrib. (4290)**

- **GH**: **Type of offset**
    - **0: Without offset**
    - **1: From left**
    - **2: From right**
    - **3: Alternating left/right**
- **A**: **Approach ang.** (range: −60° < **A** < 60°; default: 30°)
    - **A** < 0: Infeed on left thread flank
    - **A** > 0: Infeed on right thread flank
- **R**: **Rem. cut depth** (only with **GV** = 4; default: 1/100 mm)
- **Q**: **Number no-load.**
- **MT**: **M after T**: **M** function that is executed after the **T** tool call
- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step
- **MFE**: **M at end**: **M** function that is executed at the end of the machining step
- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
    - Main drive
    - Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)

> **i** Type of machining for technology database access: **Thread cutting**

Parameter combinations for the taper angle:

- **X1/Z1**, **X2/Z2**
- **X1/Z1**, **Z2**, **W**
- **Z1**, **X2/Z2**, **W**

Cycle run:

1 Calculate the proportioning of cuts
2 Move to the **Start point X1**, **Z1**
3 Move to **End point thread Z2** at programmed feed rate, taking the **Run-out angle** into account **WE**
4 Return along a paraxial path and approach for the next thread groove
5 Repeat 3 to 4 for all thread grooves
6 Approach for the next pass, taking the **reduced cutting depth** and the **Approach ang. A** into account
7 Repeat 3 to 6 until **No.gears D** and **Thread depth U** are reached
8 Move to the **Tool change point** according to the setting in **G14**

Recutting threads (longitudinal) (option 11)

▶ Select **Thread cutting**

▶ Select **Thread cycle**
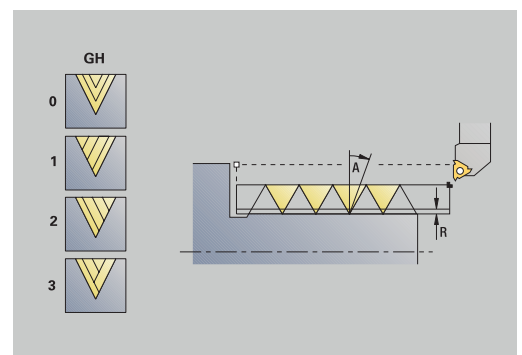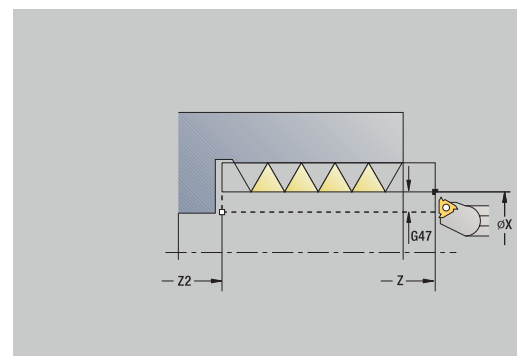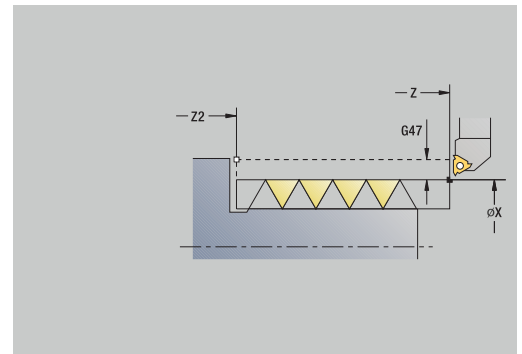
Re-cut

▶ Press the **Recut** soft key

Inner thread

▶ Select the type of thread:
  - **On:** Internal thread
  - **Off:** External thread

This optional cycle recuts a single-start thread. Since you have already unclamped the workpiece, the control needs to know the exact position of the thread. Place the cutting tip of the threading tool in the center of a groove and transfer the positions to the **Measured angle C** and **Measured position ZC** parameters by pressing the **Take over position** soft key. From these values, the cycle then calculates the angle of the spindle at the starting point.

This function is only available in **Machine** operating mode.

Cycle parameters:

- **X**, **Z**: **Start point**
- **X1**: **Start point thread**
- **Z2**: **End point thread**
- **F1**: **Thread pitch** (= feed rate)
- **U**: **Thread depth** (default: no input)
  - Outside thread: **U** = 0.6134 * **F1**
  - Inside thread: **U** = −0.5413 * **F1**
- **I**: **Max. approach**
  - **I** < **U**: First cut with **I**, all subsequent cuts with cutting-depth reduction
  - **I** = **U**: One cut
  - No input: **I** is calculated from **U** and **F1**
- **C**: **Measured angle**
- **ZC**: **Measured position**
- **T**: **Tool number** – turret pocket number
- **ID**: **ID no.**
- **S**: **Constant spindle speed**
- **GV**: **Type of infeed**
  Further information: "GV parameter: Type of infeed", Page 335
  - **0: Const. mach. X-section**
  - **1: Const. infeed**
  - **2: EPL with distrib. of cuts**
  - **3: EPL w/o distrib. of cuts**
  - **4: MANUALplus 4110**
  - **5: Constant infeed (4290)**
  - **6: Const. w/ distrib. (4290)**
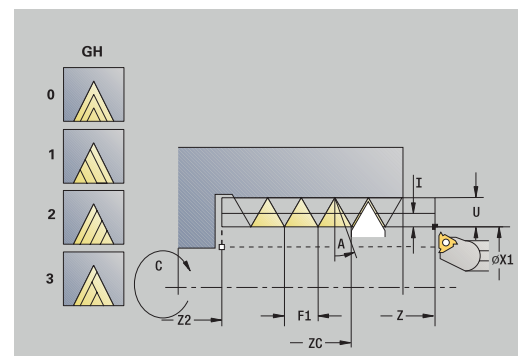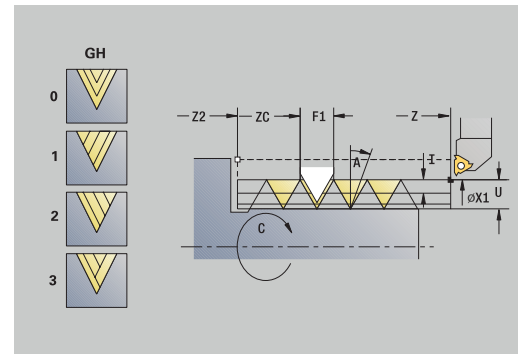
- **GH**: **Type of offset**
  - **0: Without offset**
  - **1: From left**
  - **2: From right**
  - **3: Alternating left/right**
- **A**: **Approach ang.** (range: −60° < **A** < 60°; default: 30°)
  - **A** < 0: Infeed on left thread flank
  - **A** > 0: Infeed on right thread flank
- **R**: **Rem. cut depth** (only with **GV** = 4; default: 1/100 mm)
- **MT**: **M after T**: **M** function that is executed after the **T** tool call
- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step
- **MFE**: **M at end**: **M** function that is executed at the end of the machining step
- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
  - Main drive
  - Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)

Cycle run:
1 Pre-position the threading tool to the center of a thread groove
2 Use the **Take over position** soft key to transfer the tool position and the spindle angle to the parameters **Measured position ZC** and **Measured angle C**
3 Retract the tool manually from the thread groove
4 Position the tool to the **Start point**
5 Start the cycle with the **Input finished** soft key, then press the **NC start** key

## Recutting threads, expanded (longitudinal) (option 11)

▶ Select **Thread cutting**

▶ Select **Thread cycle**

Expanded

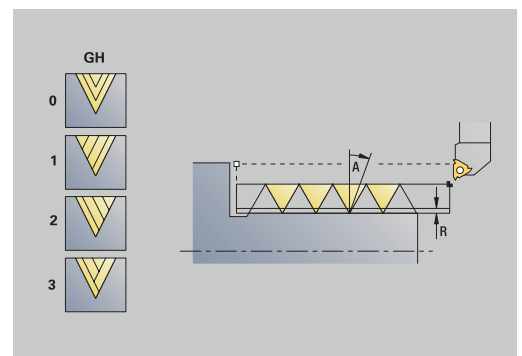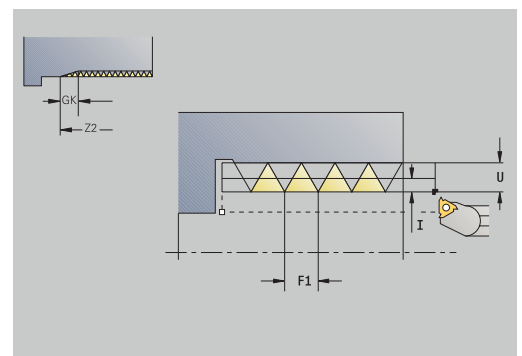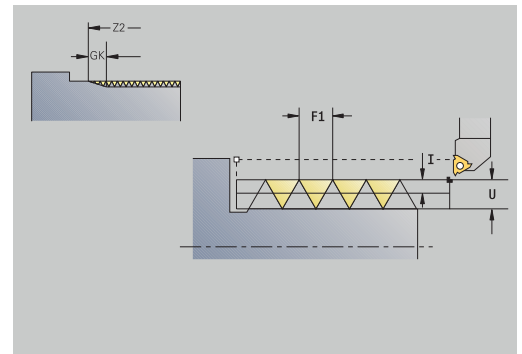▶ Select the **Expanded** soft key

Re-
cut

▶ Press the **Recut** soft key

Inner
thread

▶ Select the type of thread:
- **On:** Internal thread
- **Off:** External thread

This optional cycle recuts a single-start thread. Since you have already unclamped the workpiece, the control needs to know the exact position of the thread. Place the cutting tip of the threading tool in the center of a groove and transfer the positions to the **Measured angle C** and **Measured position ZC** parameters by pressing the **Take over position** soft key. From these values, the cycle then calculates the angle of the spindle at the starting point.

This function is only available in **Machine** operating mode.

Cycle parameters:
- **X**, **Z**: **Start point**
- **X1**: **Start point thread**
- **Z2**: **End point thread**
- **F1**: **Thread pitch** (= feed rate)
- **D**: **No.gears** (default: 1 single-start thread)
- **U**: **Thread depth** (default: no input)
  - Outside thread: **U** = 0.6134 * **F1**
  - Inside thread: **U** = −0.5413 * **F1**
- **I**: **Max. approach**
  - **I** < **U**: First cut with **I**, all subsequent cuts with cutting-depth reduction
  - **I** = **U**: One cut
  - No input: **I** is calculated from **U** and **F1**
- **GK**: **Thread runout length**
- **C**: **Measured angle**
- **ZC**: **Measured position**
- **T**: **Tool number** – turret pocket number
- **ID**: **ID no.**
- **S**: **Constant spindle speed**

- **GV**: **Type of infeed**
  Further information: "GV parameter: Type of infeed", Page 335
  - **0: Const. mach. X-section**
  - **1: Const. infeed**
  - **2: EPL with distrib. of cuts**
  - **3: EPL w/o distrib. of cuts**
  - **4: MANUALplus 4110**
  - **5: Constant infeed (4290)**
  - **6: Const. w/ distrib. (4290)**
- **GH**: **Type of offset**
  - **0: Without offset**
  - **1: From left**
  - **2: From right**
  - **3: Alternating left/right**
- **A**: **Approach ang.** (range: −60° < **A** < 60°; default: 30°)
  - **A** < 0: Infeed on left thread flank
  - **A** > 0: Infeed on right thread flank
- **R**: **Rem. cut depth** (only with **GV** = 4; default: 1/100 mm)
- **E**: **Incremental gradient** – Variable thread pitch (e.g., for manufacturing spiral conveyors or extruder shafts)
- **Q**: **Number no-load.**
- **MT**: **M after T**: **M** function that is executed after the **T** tool call
- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step
- **MFE**: **M at end**: **M** function that is executed at the end of the machining step
- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
  - Main drive
  - Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)

Cycle run:

1 Pre-position the threading tool to the center of a thread groove
2 Use the **Take over position** soft key to transfer the tool position and the spindle angle to the parameters **Measured position ZC** and **Measured angle C**
3 Retract the tool manually from the thread groove
4 Position the tool to the **Start point**
5 Start the cycle with the **Input finished** soft key and press the **NC start** key

Recutting taper threads (option 11)
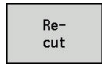
▶ Select **Thread cutting**

▶ Select **Taper thread**
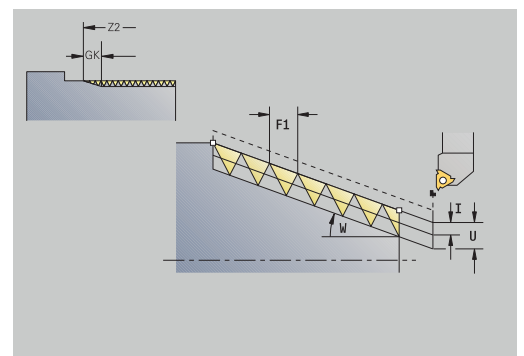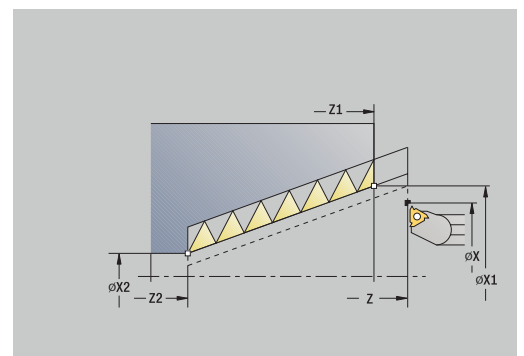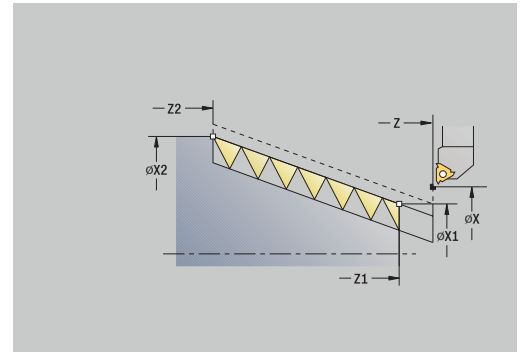
▶ Press the **Recut** soft key
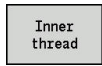
▶ Select the type of thread:
  - **On:** Internal thread
  - **Off:** External thread

This optional cycle recuts a single or multi-start tapered external or internal thread. Since you have already unclamped the workpiece, the control needs to know the exact position of the thread. Place the cutting tip of the threading tool in the center of a groove and transfer the positions to the **Measured angle C** and **Measured position ZC** parameters by pressing the **Take over position** soft key. From these values, the cycle then calculates the angle of the spindle at the starting point.

This function is only available in **Machine** operating mode.

Cycle parameters:
- **X**, **Z**: **Start point**
- **X1**, **Z1**: **Start point thread**
- **X2**, **Z2**: **End point thread**
- **F1**: **Thread pitch** (= feed rate)
- **D**: **No.gears** (default: 1 single-start thread)
- **U**: **Thread depth** (default: no input)
  - Outside thread: **U** = 0.6134 * **F1**
  - Inside thread: **U** = −0.5413 * **F1**
- **I**: **Max. approach**
  - **I** < **U**: First cut with **I**, all subsequent cuts with cutting-depth reduction
  - **I** = **U**: One cut
  - No input: **I** is calculated from **U** and **F1**
- **W**: **Taper angle** (range: −60° < **W** < 60°)
- **GK**: **Thread runout length**
  - **GK** < 0: Run-out at beginning of thread
  - **GK** > 0: Run-out at end of thread
- **C**: **Measured angle**
- **ZC**: **Measured position**
- **T**: **Tool number** – turret pocket number
- **ID**: **ID no.**
- **S**: **Constant spindle speed**

- ■ **GV**: **Type of infeed**
  Further information: "GV parameter: Type of infeed", Page 335
  - ■ **0: Const. mach. X-section**
  - ■ **1: Const. infeed**
  - ■ **2: EPL with distrib. of cuts**
  - ■ **3: EPL w/o distrib. of cuts**
  - ■ **4: MANUALplus 4110**
  - ■ **5: Constant infeed (4290)**
  - ■ **6: Const. w/ distrib. (4290)**
- ■ **GH**: **Type of offset**
  - ■ **0: Without offset**
  - ■ **1: From left**
  - ■ **2: From right**
  - ■ **3: Alternating left/right**
- ■ **A**: **Approach ang.** (range: −60° < **A** < 60°; default: 30°)
  - ■ **A** < 0: Infeed on left thread flank
  - ■ **A** > 0: Infeed on right thread flank
- ■ **R**: **Rem. cut depth** (only with **GV** = 4; default: 1/100 mm)
- ■ **E**: **Incremental gradient** – Variable thread pitch (e.g., for manufacturing spiral conveyors or extruder shafts)
- ■ **Q**: **Number no-load.**
- ■ **MT**: **M after T**: **M** function that is executed after the **T** tool call
- ■ **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step
- ■ **MFE**: **M at end**: **M** function that is executed at the end of the machining step
- ■ **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
  - ■ Main drive
  - ■ Opposing spindle for rear-face machining
- ■ **BW**: **Angle in the B axis** (machine-dependent)
- ■ **CW**: **Reverse the tool** (machine-dependent)
- ■ **HC**: **Shoe brake** (machine-dependent)
- ■ **DF**: **Miscellaneous function** (machine-dependent)

Cycle run:

1 Pre-position the threading tool to the center of a thread groove
2 Use the **Take over position** soft key to transfer the tool position and the spindle angle to the parameters **Measured position ZC** and **Measured angle C**
3 Retract the tool manually from the thread groove
4 Position the tool **in front of** the workpiece
5 Start the cycle with the **Input finished** soft key, and press the **NC start** key

Recutting API threads (option 11)
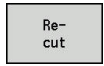
▶ Select **Thread cutting**
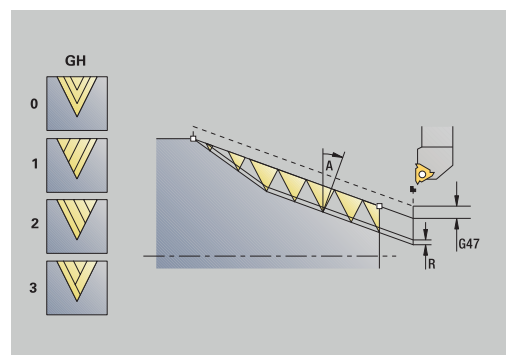
▶ Select **API thread**
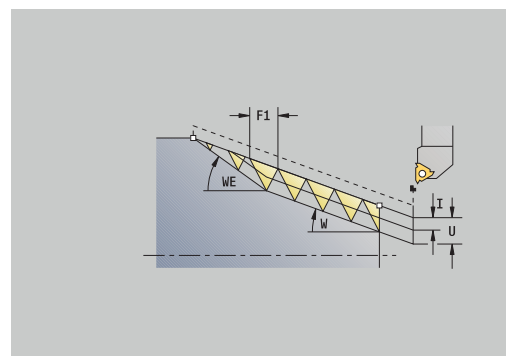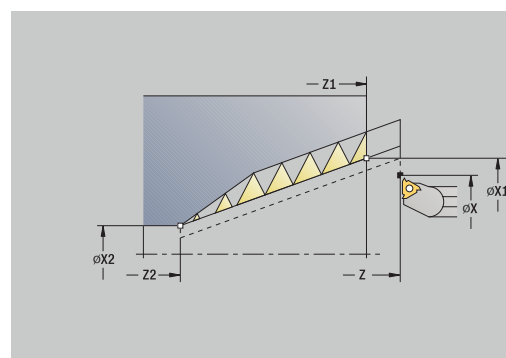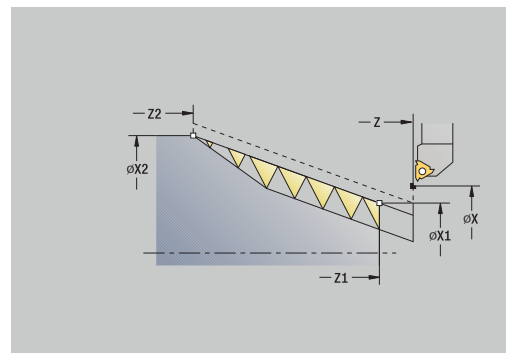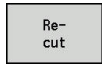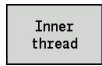
▶ Press the **Recut** soft key
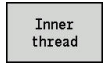
▶ Select the type of thread:
- **On:** Internal thread
- **Off:** External thread

This optional cycle recuts a single or multi-start external or internal API thread. Since you have already unclamped the workpiece, the control needs to know the exact position of the thread. Place the cutting tip of the threading tool in the center of a groove and transfer the positions to the **Measured angle C** and **Measured position ZC** parameters by pressing the **Take over position** soft key. From these values, the cycle then calculates the angle of the spindle at the starting point.

This function is only available in **Machine** operating mode.

Cycle parameters:

- **X**, **Z**: **Start point**
- **X1**, **Z1**: **Start point thread**
- **X2**, **Z2**: **End point thread**
- **F1**: **Thread pitch** (= feed rate)
- **D**: **No.gears** (default: 1 single-start thread)
- **U**: **Thread depth** (default: no input)
  - Outside thread: **U** = 0.6134 * **F1**
  - Inside thread: **U** = −0.5413 * **F1**
- **I**: **Max. approach**
  - **I** < **U**: First cut with **I**, all subsequent cuts with cutting-depth reduction
  - **I** = **U**: One cut
  - No input: **I** is calculated from **U** and **F1**
- **WE**: **Run-out angle** (range: 0° < **WE** < 90°)
- **W**: **Taper angle** (range: −60° < **W** < 60°)
- **C**: **Measured angle**
- **ZC**: **Measured position**
- **T**: **Tool number** – turret pocket number
- **ID**: **ID no.**
- **S**: **Constant spindle speed**

- **GV**: **Type of infeed**
  Further information: "GV parameter: Type of infeed", Page 335
  - **0: Const. mach. X-section**
  - **1: Const. infeed**
  - **2: EPL with distrib. of cuts**
  - **3: EPL w/o distrib. of cuts**
  - **4: MANUALplus 4110**
  - **5: Constant infeed (4290)**
  - **6: Const. w/ distrib. (4290)**
- **GH**: **Type of offset**
  - **0: Without offset**
  - **1: From left**
  - **2: From right**
  - **3: Alternating left/right**
- **A**: **Approach ang.** (range: −60° < **A** < 60°; default: 30°)
  - **A** < 0: Infeed on left thread flank
  - **A** > 0: Infeed on right thread flank
- **R**: **Rem. cut depth** (only with **GV** = 4; default: 1/100 mm)
- **Q**: **Number no-load.**
- **MT**: **M after T**: **M** function that is executed after the **T** tool call
- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step
- **MFE**: **M at end**: **M** function that is executed at the end of the machining step
- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
  - Main drive
  - Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)

Cycle run:

1 Pre-position the threading tool to the center of a thread groove
2 Use the **Take over position** soft key to transfer the tool position and the spindle angle to the parameters **Measured position ZC** and **Measured angle C**
3 Retract the tool manually from the thread groove
4 Position the tool **in front of** the workpiece
5 Start the cycle with the **Input finished** soft key, and press the **NC start** key

Undercut DIN 76

▶ Select **Thread cutting**

▶ **Undercut DIN 76**

▶ Select the scope of the cycle:
  - **Off**: When the cycle is completed, the tool remains at the cycle end position
  - **On**: The tool returns to the starting point

The cycle machines an **Undercut DIN 76**, a thread chamfer, the adjoining cylinder, and then the plane surface. The thread chamfer is executed when you enter at least one of the parameters **Cylinder 1st cut length** or **Cut-in radius**.

Cycle parameters:
- **X**, **Z**: **Start point**
- **X1**, **Z1**: **Start point cylinder**
- **X2**, **Z2**: **End point transv.**
- **FP**: **Thread pitch** (default: value from standard table)
- **E**: **Reduced feed** for the plunge cut and the thread chamfer (default: **Feed per revolution F**)
- **I**: **Undercut depth** (default: value from standard table)
- **K**: **Undercut length** (default: standard table)
- **W**: **Undercut angle** (default: value from standard table)
- **R**: **Undercut radius** on both sides of the undercut (default: value from standard table)
- **P1**: **Undercut oversize**
  - No input: Machining in one cut
  - **P1** > 0: Division into pre-turning and finish-turning. **P1** is longitudinal oversize; the transverse oversize is preset to 0.1 mm
- **T**: **Tool number** – turret pocket number
- **G14**: **Tool change point**
  Further information: "Tool change point G14", Page 203
- **ID**: **ID no.**
- **S**: **Cutting speed** or **Constant speed**
- **F**: **Feed per revolution**
- **B**: **Cylinder 1st cut length** (default: no start chamfer)
- **WB**: **Cut-in angle** (default: 45°)
- **RB**: **Cut-in radius** (no input: no element; positive value: cut-in radius; negative value: chamfer)
- **G47**: **Safety clearance**
  Further information: "Safety clearance G47", Page 203 – evaluated only if "With return" is active
- **MT**: **M after T**: **M** function that is executed after the **T** tool call
- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step
- **MFE**: **M at end**: **M** function that is executed at the end of the machining step

- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
  - Main drive
  - Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)

> **i** Type of machining for technology database access:
> **Finishing**

All parameters that you enter will be accounted for, even if the standard table prescribes other values. If the parameters **I**, **K**, **W** and **R** are not defined, the control determines these parameters from **FP** in the standard table.

Further information: "DIN 76 – undercut parameters", Page 779

Cycle run:

1 Approach the workpiece from the **Start point**
   - to the **Start point cylinder X1**
     Alternative:
   - for the **thread chamfer**
2 Machine the **thread chamfer**, if defined
3 Finish-machine the cylinder up to the beginning of the undercut
4 Pre-machine the undercut, if defined
5 Machine the undercut
6 Finish-machine to **End point transv. X2**
7 Return
   - Without return: Tool remains at the **End point transv.**
   - With return: Retract and return diagonally to the **Start point**
8 Move to the **Tool change point** according to the setting in **G14**

Undercut DIN 509 E

▶ Select **Thread cutting**

▶ **Undercut DIN 509 E**
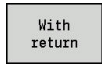
▶ Select the scope of the cycle:
  - **Off**: When the cycle is completed, the tool remains at the cycle end position
  - **On**: The tool returns to the starting point

The cycle machines an **Undercut DIN 509 E**, a cylinder start chamfer, the adjoining cylinder, and then the plane surface. You can define a grinding oversize for the area of the cylinder. The cylinder chamfer is executed when you enter at least one of the parameters **Cylinder 1st cut length** or **Cut-in radius**.

Cycle parameters:

- **X**, **Z**: **Start point**
- **X1**, **Z1**: **Start point cylinder**
- **X2**, **Z2**: **End point transv.**
- **U**: **Grind. overs.** for the area of the cylinder (default: 0)
- **E**: **Reduced feed** for the plunge cut and the thread chamfer (default: **Feed per revolution F**)
- **I**: **Undercut depth** (default: value from standard table)
- **K**: **Undercut length** (default: standard table)
- **W**: **Undercut angle** (default: value from standard table)
- **R**: **Undercut radius** on both sides of the undercut (default: value from standard table)
- **T**: **Tool number** – turret pocket number
- **G14**: **Tool change point**
  Further information: "Tool change point G14", Page 203
- **ID**: **ID no.**
- **S**: **Cutting speed** or **Constant speed**
- **F**: **Feed per revolution**
- **B**: **Cylinder 1st cut length** (default: no start chamfer)
- **WB**: **Cut-in angle** (default: 45°)
- **RB**: **Cut-in radius** (no input: no element; positive value: cut-in radius; negative value: chamfer)
- **G47**: **Safety clearance**
  Further information: "Safety clearance G47", Page 203 – evaluated only if "With return" is active
- **MT**: **M after T**: **M** function that is executed after the **T** tool call

- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step
- **MFE**: **M at end**: **M** function that is executed at the end of the machining step
- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
    - Main drive
    - Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)

> **ℹ** Type of machining for technology database access:
> **Finishing**

All parameters that you enter will be accounted for, even if the standard table prescribes other values. If the parameters **I**, **K**, **W** and **R** are not defined, the control determines these parameters from **FP** in the standard table.

Further information: "DIN 509 E – undercut parameters", Page 780

Cycle run:

1 Approach the workpiece from the **Start point**
   - to the **Start point cylinder X1**
     Alternative:
   - for the **thread chamfer**
2 Machine the **thread chamfer**, if defined
3 Finish-machine the cylinder up to the beginning of the undercut
4 Machine the undercut
5 Finish-machine to **End point transv. X2**
6 Return
   - Without return: Tool remains at the **End point transv.**
   - With return: Retract and return diagonally to the **Start point**
7 Move to the **Tool change point** according to the setting in **G14**

Undercut DIN 509 F

▶ Select **Thread cutting**

▶ **Undercut DIN 509 F**

▶ Select the scope of the cycle:
- **Off**: When the cycle is completed, the tool remains at the cycle end position
- **On**: The tool returns to the starting point

The cycle machines an **Undercut DIN 509 F**, a cylinder start chamfer, the adjoining cylinder, and then the plane surface. You can define a grinding oversize for the area of the cylinder. The cylinder chamfer is executed when you enter at least one of the parameters **Cylinder 1st cut length** or **Cut-in radius**.

Cycle parameters:
- **X**, **Z**: **Start point**
- **X1**, **Z1**: **Start point cylinder**
- **X2**, **Z2**: **End point transv.**
- **U**: **Grind. overs.** for the area of the cylinder (default: 0)
- **E**: **Reduced feed** for the plunge cut and the thread chamfer (default: **Feed per revolution F**)
- **I**: **Undercut depth** (default: value from standard table)
- **K**: **Undercut length** (default: standard table)
- **W**: **Undercut angle** (default: value from standard table)
- **R**: **Undercut radius** on both sides of the undercut (default: value from standard table)
- **P2**: **Surf. depth** (default: value from standard table)
- **A**: **Face angle** (default: standard table)
- **T**: **Tool number** – turret pocket number
- **G14**: **Tool change point**
  Further information: "Tool change point G14", Page 203
- **ID**: **ID no.**
- **S**: **Cutting speed** or **Constant speed**
- **F**: **Feed per revolution**
- **B**: **Cylinder 1st cut length** (default: no start chamfer)
- **WB**: **Cut-in angle** (default: 45°)
- **RB**: **Cut-in radius** (no input: no element; positive value: cut-in radius; negative value: chamfer)
- **G47**: **Safety clearance**
  Further information: "Safety clearance G47", Page 203 – evaluated only if **With return** is active
- **MT**: **M after T**: **M** function that is executed after the **T** tool call

- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step
- **MFE**: **M at end**: **M** function that is executed at the end of the machining step
- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
  - Main drive
  - Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)

> **i** Type of machining for technology database access:
> **Finishing**

All parameters that you enter will be accounted for, even if the standard table prescribes other values. If the parameters **I**, **K**, **W** and **R** are not defined, the control determines these parameters from **FP** in the standard table.

Further information: "", Page 780

Cycle run:

1 Approach the workpiece from the **Start point**
  - to the **Start point cylinder X1**
    Alternative:
  - for the **thread chamfer**
2 Machine the **thread chamfer**, if defined
3 Finish-machine the cylinder up to the beginning of the undercut
4 Machine the undercut
5 Finish-machine to **End point transv. X2**
6 Return
  - Without return: Tool remains at the **End point transv.**
  - With return: Retract and return diagonally to the **Start point**
7 Move to the **Tool change point** according to the setting in **G14**

Example of thread and undercut cycles

External thread and thread undercut

The machining operation consists of two steps. The **Undercut DIN 76** cycle machines the undercut and thread chamfer. The **thread cycle** then cuts the thread.
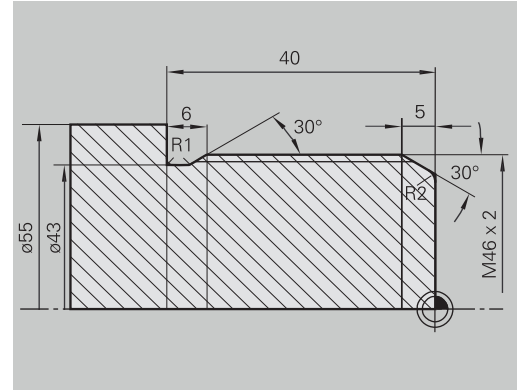
1st step

The parameters for the undercut and thread chamfer can be programmed in two superimposed input windows.

Tool data:

- Turning tool (for external machining)
- **TO** = 1 – Tool orientation
- **A** = 93° – Tool angle
- **B** = 55° – Point angle
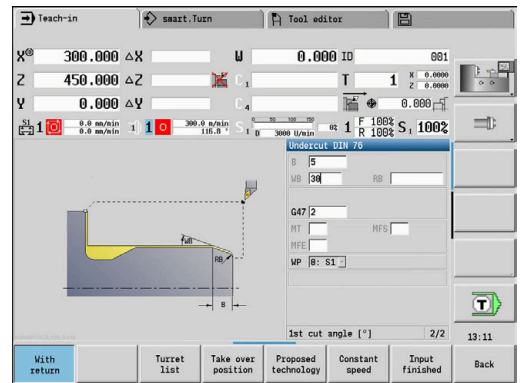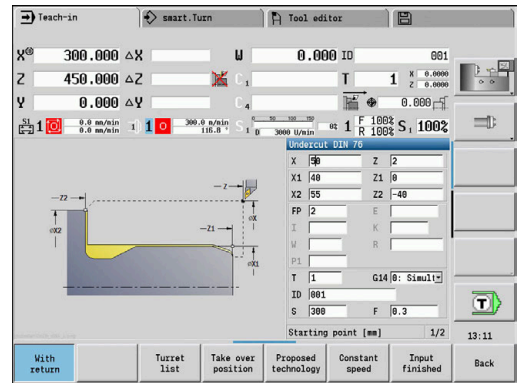
2nd step

The **thread cycle (longitudinal)—expanded** cuts the thread. The cycle parameters define the thread depth and the proportioning of cuts.

Tool data:

- Turning tool (for external machining)
- **TO** = 1 – Tool orientation
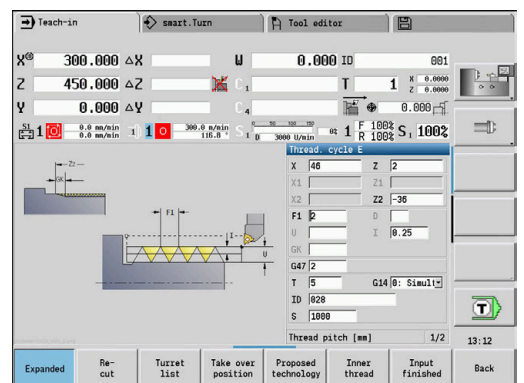
Internal thread and thread undercut

The machining operation consists of two steps. The **Undercut DIN 76** cycle machines the undercut and thread chamfer. The **thread cycle** then cuts the thread.
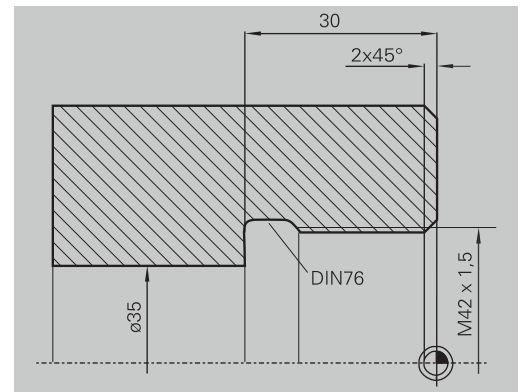
1st step

The parameters for the undercut and thread chamfer can be programmed in two superimposed input windows.

The control determines the undercut parameters from the standard table.

For the thread chamfer, you only need to enter the chamfer width. The angle of 45° is the default value for the **Cut-in angle WB**.

Tool data:

- Turning tool (for internal machining)
- **TO** = 7 – Tool orientation
- **A** = 93° – Tool angle
- **B** = 55° – Point angle
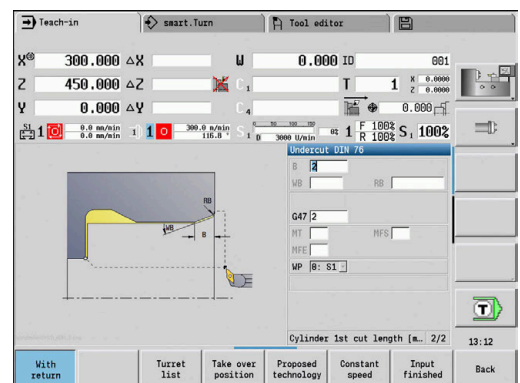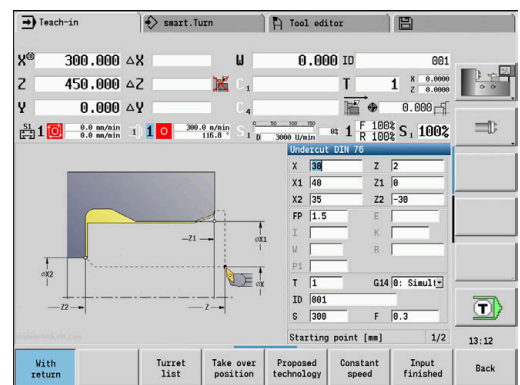
2nd step

The **thread cycle (longitudinal)** cuts the thread. The thread pitch is defined. The control automatically determines all other values from the standard table.

You must pay attention to the setting of the **Inner thread** soft key.

Tool data:

- Turning tool (for internal machining)
- **TO** = 7 – Tool orientation
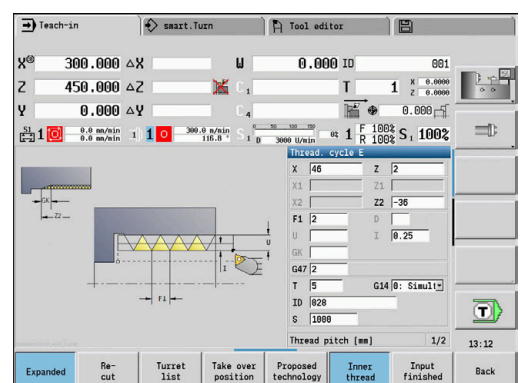
7.7 Drilling cycles

Menu item	Meaning
	The drilling cycles allow you to machine axial and radial holes

Pattern machining:
Further information: "Drilling and milling patterns", Page 418

Menu item		Drilling cycles
		Drill. axial/Drill. radial For drilling single holes and patterns
		Deep hole ax./Deep hole rad For drilling single holes and patterns
		Tapping axial/Tapping radial For drilling single holes and patterns
		Thread milling axial For milling threads in existing holes

Drill. axial

▶ Select **Drilling**

▶ Select **Drilling axial**

This cycle drills a hole on the face of the workpiece.

Cycle parameters:

- **X**, **Z**: **Start point**
- **C**: **Spindle angle** – C-axis position
- **Z1**: **Start point drill** (default: drilling starts from **Z**)
- **Z2**: **End point drill**
- **E**: **Period of dwell** for chip breaking at end of hole (default: 0)
- **D**: **Retraction type**
  - **0: Rapid traverse**
  - **1: Feed rate**
- **AB**: **Drilling lengths** (default: 0)
- **V**: **Drilling variants** (default: 0)
  - **0: Without reduction**
  - **1: At end of the hole**
  - **2: At start of the hole**
  - **3: At start & end of hole**
- **CB**: **Brake off (1)**
- **SCK**: **Safety clearance**
  Further information: "Safety clearances SCI and SCK", Page 203

- **G60**: Deactivate **Protection zone** for the drilling operation
    - **0: Active**
    - **1: Inactive**
- **T**: **Tool number** – turret pocket number
- **G14**: **Tool change point**
  Further information: "Tool change point G14", Page 203
- **ID**: **ID no.**
- **S**: **Cutting speed** or **Constant speed**
- **F**: **Feed per revolution**
- **BP**: **Interval time** – time during which the feed motion will be interrupted
  The chip is broken by the intermittent feed.
- **BF**: **Feed period** – time interval until the next break
  The chip is broken by the intermittent feed.
- **MT**: **M after T**: **M** function that is executed after the **T** tool call
- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step
- **MFE**: **M at end**: **M** function that is executed at the end of the machining step
- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
    - Main drive
    - Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)

> 🛈 Type of machining for technology database access depends on the tool type:
> - **Drillg** for twist drills
> - **Predrilling** with indexable insert drills

> 🛈 - If **AB** and **V** are programmed, the feed rate is reduced by 50 % during both pre-drilling and through-boring.
> - The control uses the tool parameter **driven tool** to determine whether the programmed spindle speed and feed rate apply to the spindle or the driven tool.

Cycle run:

1 Position the spindle to the **Spindle angle C** (in **Machine** operating mode: machining from current spindle angle)

2 If defined, move at rapid traverse to the **Start point drill Z1**

3 If defined, spot drill at reduced feed rate

4 Depending on **Drilling variants V**:
- Through-drilling reduction:
  - Drill at programmed feed rate to position **Z2 − AB**
  - Drill at reduced feed rate to **End point drill Z2**
- No through-drilling reduction:
  - Drill at programmed feed rate to **End point drill Z2**
  - Remain at end of hole for **Period of dwell E**, if defined

5 Retract
- If **Z1** is programmed, to **Start point drill Z1**
- If **Z1** is not programmed, to **Start point Z**

6 Move to the **Tool change point** according to the setting in **G14**

Drilling radial

▶ Select **Drilling**

▶ Select **Drilling radial**

This cycle drills a hole on the lateral surface of the workpiece.
Cycle parameters:

- **X**, **Z**: **Start point**
- **C**: **Spindle angle** – C-axis position
- **X1**: **Start point drill** (default: drilling starts from **X**)
- **X2**: **End point drill**
- **E**: **Period of dwell** for chip breaking at end of hole (default: 0)
- **D**: **Retraction type**
    - **0: Rapid traverse**
    - **1: Feed rate**
- **AB**: **Drilling lengths** (default: 0)
- **V**: **Drilling variants** (default: 0)
    - **0: Without reduction**
    - **1: At end of the hole**
    - **2: At start of the hole**
    - **3: At start & end of hole**
- **CB**: **Brake off (1)**
- **SCK**: **Safety clearance**
  Further information: "Safety clearances SCI and SCK", Page 203
- **T**: **Tool number** – turret pocket number
- **G14**: **Tool change point**
  Further information: "Tool change point G14", Page 203
- **ID**: **ID no.**
- **S**: **Cutting speed** or **Constant speed**
- **F**: **Feed per revolution**
- **BP**: **Interval time** – time during which the feed motion will be interrupted
  The chip is broken by the intermittent feed.
- **BF**: **Feed period** – time interval until the next break
  The chip is broken by the intermittent feed.
- **MT**: **M after T**: **M** function that is executed after the **T** tool call

- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step
- **MFE**: **M at end**: **M** function that is executed at the end of the machining step
- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
    - Main drive
    - Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)

> Type of machining for technology database access depends on the tool type:
>   - **Drillg** for twist drills
>   - **Predrilling** with indexable insert drills

> If **AB** and **V** are programmed, the feed rate is reduced by 50 % during both pre-drilling and through-boring.

Cycle run:
1. Position the spindle to the **Spindle angle C** (in **Machine** operating mode: machining from current spindle angle)
2. If defined, move at rapid traverse to the **Start point drill X1**
3. If defined, spot drill at reduced feed rate
4. Depending on **Drilling variants V**:
    - Through-drilling reduction:
        - Drill at programmed feed rate to position **X2** – **AB**
        - Drill at reduced feed rate to **End point drill X2**
    - No through-drilling reduction:
        - Drill at programmed feed rate to **End point drill X2**
        - Remain at end of hole for **Period of dwell E**, if defined
5. Retract
    - If **X1** is programmed, to **Start point drill X1**
    - If **X1** is not programmed, to **Start point X**
6. Move to the **Tool change point** according to the setting in **G14**

Deep drilling axial

▶ Select **Drilling**

▶ Select **Deep drilling axial**

The cycle produces a hole on a front face in several passes.

Cycle parameters:

- **X**, **Z**: **Start point**
- **C**: **Spindle angle** – C-axis position
- **Z1**: **Start point drill** (default: drilling starts from **Z**)
- **Z2**: **End point drill**
- **P**: **1st hole dpth** (default: hole will be drilled in one pass)
- **IB**: **Hole depth reduct. val** (default: 0)
- **JB**: **Minimum hole depth** (default: 1/10 of **P**)
- **B**: **Return length** (default: retract to **Start point drill**)
- **E**: **Period of dwell** for chip breaking at end of hole (default: 0)
- **D**: **Retraction type** – Retraction speed and infeed within the hole (default: 0)
  - **0: Rapid traverse**
  - **1: Feed rate**
- **AB**: **Drilling lengths** (default: 0)
- **V**: **Drilling variants** (default: 0)
  - **0: Without reduction**
  - **1: At end of the hole**
  - **2: At start of the hole**
  - **3: At start & end of hole**
- **T**: **Tool number** – turret pocket number
- **G14**: **Tool change point**
  Further information: "Tool change point G14", Page 203
- **ID**: **ID no.**
- **S**: **Cutting speed** or **Constant speed**
- **F**: **Feed per revolution**
- **CB**: **Brake off (1)**
- **SCK**: **Safety clearance**
  Further information: "Safety clearances SCI and SCK", Page 203
- **G60**: Deactivate **Protection zone** for the drilling operation
  - **0: Active**
  - **1: Inactive**

- **BP**: **Interval time** – time during which the feed motion will be interrupted
  The chip is broken by the intermittent feed.
- **BF**: **Feed period** – time interval until the next break
  The chip is broken by the intermittent feed.
- **MT**: **M after T**: **M** function that is executed after the **T** tool call
- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step
- **MFE**: **M at end**: **M** function that is executed at the end of the machining step
- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
  - Main drive
  - Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)

> **ℹ** Type of machining for technology database access depends on the tool type:
> - **Drillg** for twist drills
> - **Predrilling** with indexable insert drills

> **ℹ**
> - If **AB** and **V** are programmed, the feed rate is reduced by 50 % during both pre-drilling and through-boring.
> - The control uses the tool parameter **driven tool** to determine whether the programmed spindle speed and feed rate apply to the spindle or the driven tool.

Cycle run:

1   Position the spindle to the **Spindle angle C** (in **Machine** operating mode: machining from current spindle angle)

2   If defined, move at rapid traverse to the **Start point drill Z1**

3   First pass (pecking depth: **P**)—spot drill at reduced feed rate, if defined

4   Retract by the **Return length B**—or to the **Start point drill** and advance again to the safety clearance inside the hole

5   Next pass (pecking depth: last depth − **IB** or **JB**)

6   Repeat 4 to 5 until the **End point drill Z2** is reached

7   Last drilling pass, depending on **Drilling variants V**:
  ■ Through-drilling reduction:
    ■ Drill at programmed feed rate to position **Z2** − **AB**
    ■ Drill at reduced feed rate to **End point drill Z2**
  ■ No through-drilling reduction:
    ■ Drill at programmed feed rate to **End point drill Z2**
    ■ Remain at end of hole for **Period of dwell E**, if defined

8   Retract
  ■ If **Z1** is programmed, to **Start point drill Z1**
  ■ If **Z1** is not programmed, to **Start point Z**

9   Move to the **Tool change point** according to the setting in **G14**

Deep drilling radial

▶ Select **Drilling**

▶ Select **Deep drilling radial**

The cycle produces a bore hole on the lateral surface of a workpiece in several passes.

Cycle parameters:

- **X**, **Z**: **Start point**
- **C**: **Spindle angle** – C-axis position
- **X1**: **Start point drill** (default: drilling starts from **X**)
- **X2**: **End point drill**
- **P**: **1st hole dpth** (default: hole will be drilled in one pass)
- **IB**: **Hole depth reduct. val** (default: 0)
- **JB**: **Minimum hole depth** (default: 1/10 of **P**)
- **B**: **Return length** (default: retract to **Start point drill**)
- **E**: **Period of dwell** for chip breaking at end of hole (default: 0)
- **D**: **Retraction type** – Retraction speed and infeed within the hole (default: 0)
  - **0: Rapid traverse**
  - **1: Feed rate**
- **AB**: **Drilling lengths** (default: 0)
- **V**: **Drilling variants** (default: 0)
  - **0: Without reduction**
  - **1: At end of the hole**
  - **2: At start of the hole**
  - **3: At start & end of hole**
- **T**: **Tool number** – turret pocket number
- **G14**: **Tool change point**
  Further information: "Tool change point G14", Page 203
- **ID**: **ID no.**
- **S**: **Cutting speed** or **Constant speed**
- **F**: **Feed per revolution**
- **CB**: **Brake off (1)**
- **SCK**: **Safety clearance**
  Further information: "Safety clearances SCI and SCK", Page 203

- **BP**: **Interval time** – time during which the feed motion will be interrupted
  The chip is broken by the intermittent feed.
- **BF**: **Feed period** – time interval until the next break
  The chip is broken by the intermittent feed.
- **MT**: **M after T**: **M** function that is executed after the **T** tool call
- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step
- **MFE**: **M at end**: **M** function that is executed at the end of the machining step
- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
  - Main drive
  - Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)

> 🛈 Type of machining for technology database access depends on the tool type:
> - **Drillg** for twist drills
> - **Predrilling** with indexable insert drills

> 🛈 If **AB** and **V** are programmed, the feed rate is reduced by 50 % during both pre-drilling and through-boring.

Cycle run:

1 Position the spindle to the **Spindle angle C** (in **Machine** operating mode: machining from current spindle angle)

2 Move at rapid traverse to **Start point drill**, if defined **X1**

3 First pass (pecking depth: **P**) – spot drill at reduced feed rate, if defined

4 Retract by the **Return length B** – or to the **Start point drill** and advance again to the safety clearance inside the hole

5 Next pass (pecking depth: last depth – **IB** or **JB**)

6 Repeat 4 to 5 until the **End point drill X2** is reached

7 Last drilling pass, depending on **Drilling variants V**:
   - Through-drilling reduction:
     - Drill at programmed feed rate to position **X2** – **AB**
     - Drill at reduced feed rate to **End point drill X2**
   - No through-drilling reduction:
     - Drill at programmed feed rate to **End point drill X2**
     - Remain at end of hole for **Period of dwell E**, if defined

8 Retract
   - If **X1** is programmed, to **Start point drill X1**
   - If **X1** is not programmed, to **Start point X**

9 Move to the **Tool change point** according to the setting in **G14**
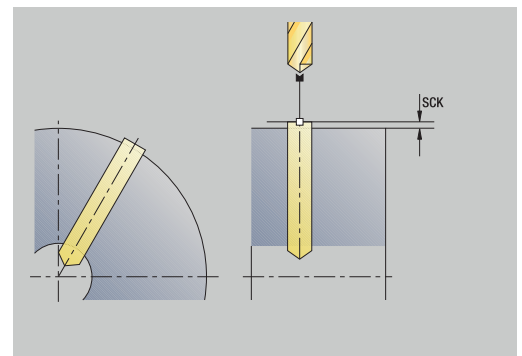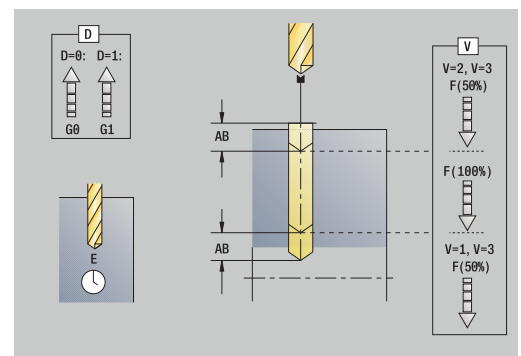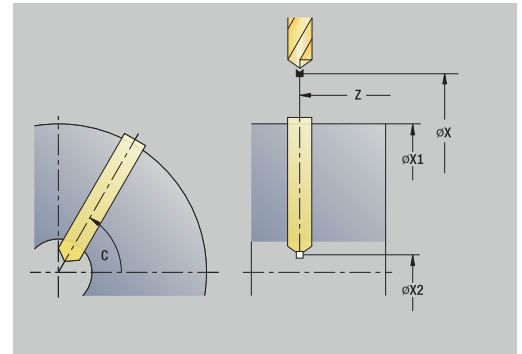
Tapping axial

▶ Select **Drilling**

▶ Select **Tapping axial**

This cycle is used to tap a thread on the face of a workpiece.

Meaning of **Retract length**: Use this parameter for floating tap holders. The cycle calculates a new nominal pitch on the basis of the thread depth, the programmed pitch, and the retraction length. The nominal pitch is somewhat smaller than the pitch of the tap. During tapping, the tap is pulled away from the chuck by the retraction length. With this method, you can achieve higher service life of the taps.

Cycle parameters:

- **X**, **Z**: **Start point**
- **C**: **Spindle angle** – C-axis position
- **Z1**: **Start point drill** (default: drilling starts from **Z**)
- **Z2**: **End point drill**
- **F1**: **Thread pitch** (default: feed rate from tool definition)
- **B**: **Run-in lgth** to obtain the programmed spindle speed and feed rate (default: 2 * **Thread pitch F1**)
- **SR**: **Return speed** for enabling rapid retraction (default: same spindle speed as for tapping)
- **L**: **Retract length** when using floating tap holders (default: 0)
- **CB**: **Brake off (1)**
- **SCK**: **Safety clearance**
  Further information: "Safety clearances SCI and SCK", Page 203
- **G60**: Deactivate **Protection zone** for the drilling operation
  - **0: Active**
  - **1: Inactive**
- **T**: **Tool number** – turret pocket number
- **G14**: **Tool change point**
  Further information: "Tool change point G14", Page 203
- **ID**: **ID no.**
- **S**: **Cutting speed** or **Constant speed**
- **SP**: **Chip breaking depth**
- **SI**: **Retraction distance**
- **MT**: **M after T**: **M** function that is executed after the **T** tool call
- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step
- **MFE**: **M at end**: **M** function that is executed at the end of the machining step

- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
  - Main drive
  - Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)

> **i** Type of machining for technology database access:
> **Tapping**

> **i** The control uses the tool parameter **driven tool** to determine whether the programmed spindle speed and feed rate apply to the spindle or the driven tool.

Cycle run:
1. Position the spindle to the **Spindle angle C** (in **Machine** operating mode: machining from current spindle angle)
2. If defined, move at rapid traverse to the **Start point drill Z1**
3. Tap the thread to the **End point drill Z2**
4. Retract at **Return speed SR**
   - If **Z1** is programmed, to **Start point drill Z1**
   - If **Z1** is not programmed, to **Start point Z**
5. Move to the **Tool change point** according to the setting in **G14**

> **i** If you interrupt program run during a tapping cycle, you can manually retract the tool from the hole in the Z axis. The control moves the spindle on a path matching the traverse.
>
> If the optional machine parameter **CfgBackTrack** (no. 122000) is active, use the **Start blck search** soft key to resume program run after the manual traverse.

Tapping radial

▶ Select **Drilling**

▶ Select **Tapping radial**

This cycle is used to tap a thread on the lateral surface of a workpiece.

Meaning of **Retract length**: Use this parameter for floating tap holders. The cycle calculates a new nominal pitch on the basis of the thread depth, the programmed pitch, and the **Retract length**. The nominal pitch is somewhat smaller than the pitch of the tap. During tapping, the drill is pulled away from the chuck by the **Retract length**. With this method, you can achieve higher service life of the taps.

Cycle parameters:

- **X**, **Z**: **Start point**
- **C**: **Spindle angle** – C-axis position
- **X1**: **Start point drill** (default: drilling starts from **X**)
- **X2**: **End point drill**
- **F1**: **Thread pitch** (default: feed rate from tool definition)
- **B**: **Run-in lgth** to obtain the programmed spindle speed and feed rate (default: 2 * **Thread pitch F1**)
- **SR**: **Return speed** for enabling rapid retraction (default: same spindle speed as for tapping)
- **L**: **Retract length** when using floating tap holders (default: 0)
- **CB**: **Brake off (1)**
- **SCK**: **Safety clearance**
  Further information: "Safety clearances SCI and SCK", Page 203
- **T**: **Tool number** – turret pocket number
- **G14**: **Tool change point**
  Further information: "Tool change point G14", Page 203
- **ID**: **ID no.**
- **S**: **Cutting speed** or **Constant speed**
- **SP**: **Chip breaking depth**
- **SI**: **Retraction distance**
- **MT**: **M after T**: **M** function that is executed after the **T** tool call
- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step
- **MFE**: **M at end**: **M** function that is executed at the end of the machining step
- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
  - Main drive
  - Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)

> ℹ️ Type of machining for technology database access:
> **Tapping**

Cycle run:

1 Position the spindle to the **Spindle angle C** (in **Machine** operating mode: machining from current spindle angle)
2 If defined, move at rapid traverse to the **Start point drill X1**
3 Tap the thread to the **End point drill X2**
4 Retract at **Return speed SR**
  - If **X1** is programmed, to **Start point drill X1**
  - If **X1** is not programmed, to **Start point X**
5 Move to the **Tool change point** according to the setting in **G14**

> ℹ️ If you interrupt program run during a tapping cycle, you can manually retract the tool from the hole in the Z axis. The control moves the spindle on a path matching the traverse.
>
> If the optional machine parameter **CfgBackTrack** (no. 122000) is active, use the **Start blck search** soft key to resume program run after the manual traverse.

Axial thread milling

▶ Select **Drilling**

▶ Select **Axial thread milling**

The cycle mills a thread in an existing hole.

ⓘ Use threading tools for this cycle.

Cycle parameters:
- **X**, **Z**: **Start point**
- **C**: **Spindle angle** – C-axis position (default: current spindle angle)
- **Z1**: **Start point drill** (default: drilling starts from **Z**)
- **Z2**: **End point drill**
- **F1**: **Thread pitch** (= feed rate)
- **J**: **Direction of thread:**
    - **0: Right-hand thread**
    - **1: Left-hand thread**
- **I**: **Thread diameter**
- **R**: **Approach radius** (default: (**I** – milling diameter)/2)
- **H**: **Mill cutting direction**
    - **0: Up-cut**
    - **1: Climb**
- **V**: **Milling method**
    - **0: One revolution** – the thread is milled in a 360-degree helix
    - **1: Two or more revolutions** – the thread is milled in several helix paths (single-point tool)
- **SCK**: **Safety clearance**
  Further information: "Safety clearances SCI and SCK", Page 203
- **T**: **Tool number** – turret pocket number
- **G14**: **Tool change point**
  Further information: "Tool change point G14", Page 203
- **ID**: **ID no.**
- **S**: **Cutting speed** or **Constant speed**
- **MT**: **M after T**: **M** function that is executed after the **T** tool call
- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step
- **MFE**: **M at end**: **M** function that is executed at the end of the machining step
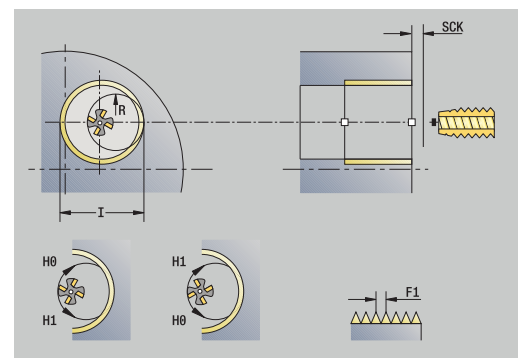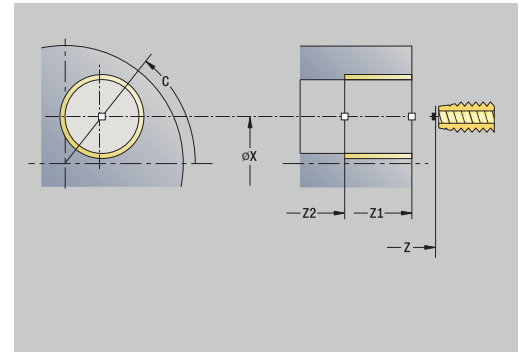
- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
  - Main drive
  - Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)

> ⓘ Type of machining for technology database access: **Milling**

Cycle run:

1 Position the spindle to the **Spindle angle C** (in **Machine** operating mode: machining from current spindle angle)
2 Position the tool to the **End point drill Z2** (milling floor) inside the hole
3 Approach on **Approach radius R**
4 Mill the thread in a rotation of 360°, while advancing by the **Thread pitch F1**
5 Retract the tool and return it to the **Start point**
6 Move to the **Tool change point** according to the setting in **G14**

Examples of drilling cycles

Centric drilling and tapping

The machining operation consists of two steps. In the first step, the **Drilling axial** cycle drills the hole. In the second step, the **Tapping axial** cycle taps the thread.

The drill is positioned at safety clearance to the workpiece surface (**Start point X**, **Z**). The **Start point drill Z1** is therefore not programmed. A feed-rate reduction for spot drilling is programmed in the **AB** and **V** parameters.

The thread pitch is not programmed. The control uses the thread pitch of the tool. The **Return speed SR** ensures that the tool is retracted quickly.

Tool data (drill)

- **TO** = 8 – Tool orientation
- **I** = 8.2 – Drilling diameter
- **B** = 118 – Point angle
- **H** = 0 – The tool is not a driven tool

Tool data (tap)

- **TO** = 8 – Tool orientation
- **I** = 10 – Thread diameter M10
- **F** = 1.5 – Thread pitch
- **H** = 0 – The tool is not a driven tool
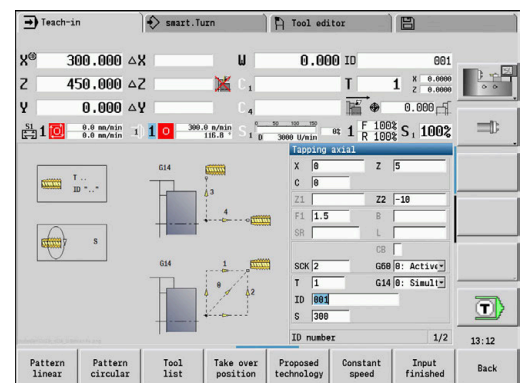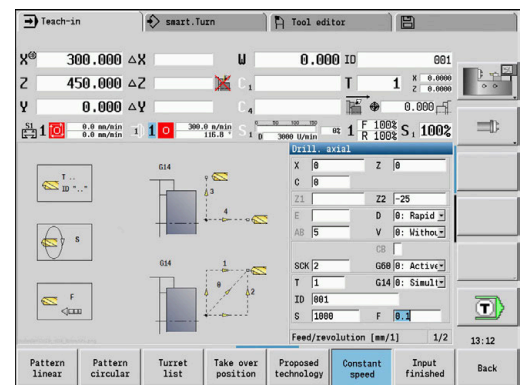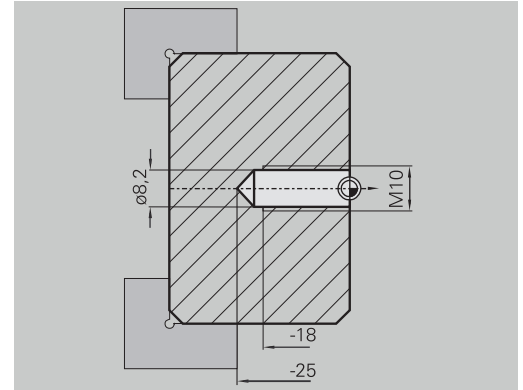
Deep-hole drilling

A through hole is drilled through the workpiece outside the turning center with the **Deep drilling axial** cycle. This machining operation requires a traversable spindle and driven tools.

The **1st hole dpth P** and **Hole depth reduct. val IB** parameters define the individual passes, and the **Minimum hole depth JB** parameter limits the hole depth reduction.

As the **Return length B** is not defined, the drill therefore retracts to the **Start point** after each pass, remains there for the programmed dwell time, and then advances again to the safety clearance for the next pass.

Since this example is to illustrate how you drill a through hole, the **End point drill Z2** is programmed such that the tool has to drill all the way through the workpiece before it reaches the end point.

The parameters **AB** and **V** define a feed-rate reduction for both spot drilling and through-boring.

Tool data

- **TO** = 8 – Tool orientation
- **I** = 12 – Drilling diameter
- **B** = 118 – Point angle
- **H** = 1 – The tool is a driven tool
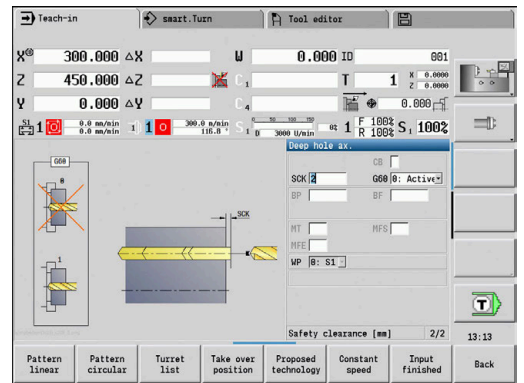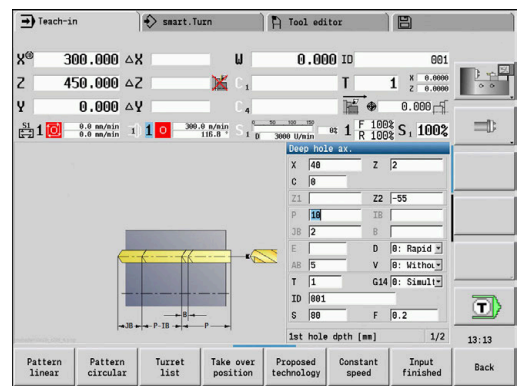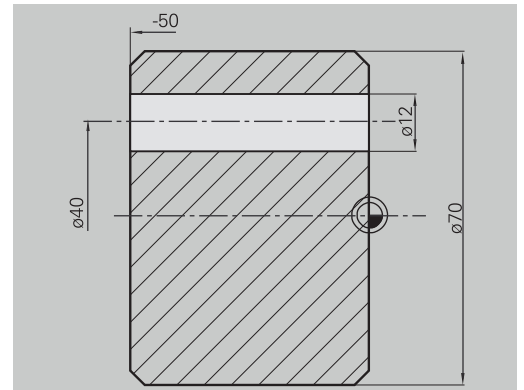
7.8 Milling cycles

Menu item	Meaning
	Milling cycles are used to machine axial and radial slots, contours, pockets, surfaces and polygons.

Pattern machining:
Further information: "Drilling and milling patterns", Page 418

In **Teach-in** submode, these cycles include activation/deactivation of the C axis and spindle positioning.

In **Machine** operating mode, activate the C axis with **Rapid positioning** to position the spindle **before** the actual milling cycle. The milling cycles then automatically deactivate the C axis.

Menu item		Milling cycles
		Rapid positioning
		Activate C axis; position tool and spindle
		Slot axial/Slot radial
		For milling single slots or slot patterns
		Figure axial/Figure radial
		For milling a single figure
		ICP contour axial/ICP contour radial
		For milling single ICP contours or contour patterns
		Face milling
		For milling surfaces or polygons
		Hel. groove millg. radial
		Radial milling for machining a helical slot
		Engraving, axial/Engraving, radial
		Engraves characters and character strings
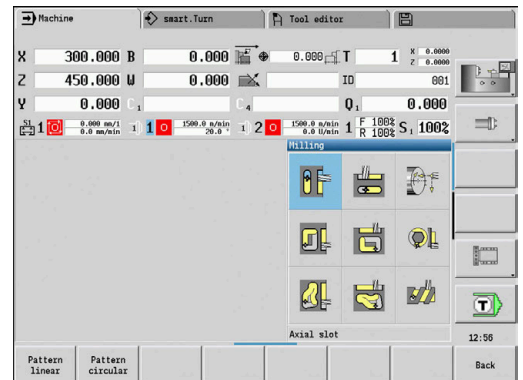
Rapid positioning, milling

▶ Select **Milling**

▶ Select **Rapid positioning**

The cycle activates the C axis and positions the spindle (C axis) and the tool.

ℹ ▪ **Rapid traverse positioning** is only possible in **Machine** operating mode
▪ The C axis is deactivated by a subsequent manual milling cycle

Cycle parameters:

- **X2**, **Z2**: **Target point**
- **C2**: **End angle** – C-axis position (default: current spindle angle)
- **MT**: **M after T**: **M** function that is executed after the **T** tool call
- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step
- **MFE**: **M at end**: **M** function that is executed at the end of the machining step
- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
  - Main drive
  - Opposing spindle for rear-face machining

Cycle run:
1 Activate the C axis
2 Insert the current tool
3 Simultaneously position the tool at rapid traverse to the **Target point X2**, **Z2**, and the **End angle C2**
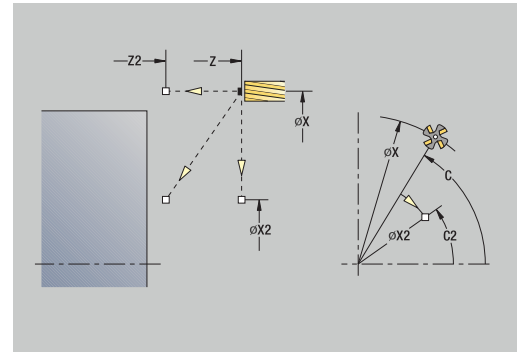
Axial slot

▶ Select **Milling**

▶ Select **Axial slot**

This cycle mills a slot on the face of the workpiece. The slot width equals the diameter of the milling cutter.

Cycle parameters:

- **X**, **Z**: **Start point**
- **C**: **Spindle angle** – C-axis position
- **X1**: **Slot target point in X** (diameter value)
- **C1**: **Angle of slot target pt.** (default: spindle angle C)
- **L**: **Slot length**
- **A1**: **Angle to X axis** (default: 0°)
- **Z1**: **Millg. top edge** (default: **Starting point Z**)
- **Z2**: **Milling floor**
- **P**: **Feed depth** (default: total depth in one infeed)
- **FZ**: **Approach feed** (default: active feed rate)
- **SCK**: **Safety clearance**
  Further information: "Safety clearances SCI and SCK", Page 203
- **T**: **Tool number** – turret pocket number
- **G14**: **Tool change point**
  Further information: "Tool change point G14", Page 203
- **ID**: **ID no.**
- **S**: **Cutting speed** or **Constant speed**
- **F**: **Feed per revolution**
- **MT**: **M after T**: **M** function that is executed after the **T** tool call
- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step
- **MFE**: **M at end**: **M** function that is executed at the end of the machining step
- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
  - Main drive
  - Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)

> **ⓘ** Type of machining for technology database access: **Milling**

Parameter combinations for the position and orientation of the slot:

- **X1**, **C1**
- **L**, **A1**

Cycle run:

1 Activate the C axis and position to **Spindle angle C** at rapid traverse (only in **Teach-in** submode)

2 Calculate the proportioning of cuts

3 Approach at **Approach feed FZ**

4 Machine to the **end point of slot**

5 Approach at **Approach feed FZ**

6 Machine to the **starting point of slot**

7 Repeat 3 to 6 until the milling depth is reached

8 Position to the **Start point Z** and deactivate the C axis

9 Move to the **Tool change point** according to the setting in **G14**

Radial slot

▶   Select **Milling**

▶   Select **Radial slot**

This cycle mills a slot on the lateral surface of the workpiece. The slot width equals the diameter of the milling cutter.

Cycle parameters:

- **X**, **Z**: **Start point**
- **C**: **Spindle angle** – C-axis position
- **Z1**: **Slot target point**
- **C1**: **Angle of slot target pt.** (default: spindle angle C)
- **L**: **Slot length**
- **A1**: **Angle to Z axis** (default: 0°)
- **X1**: **Upper edge of milling** (diameter value; default: **Starting point X**)
- **X2**: **Milling floor**
- **P**: **Feed depth** (default: total depth in one infeed)
- **FZ**: **Approach feed** (default: active feed rate)
- **SCK**: **Safety clearance**
  Further information: "Safety clearances SCI and SCK", Page 203
- **T**: **Tool number** – turret pocket number
- **G14**: **Tool change point**
  Further information: "Tool change point G14", Page 203
- **ID**: **ID no.**
- **S**: **Cutting speed** or **Constant speed**
- **F**: **Feed per revolution**
- **MT**: **M after T**: **M** function that is executed after the **T** tool call
- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step
- **MFE**: **M at end**: **M** function that is executed at the end of the machining step
- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
  - Main drive
  - Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)

> ⓘ   Type of machining for technology database access: **Milling**

Parameter combinations for the position and orientation of the slot:

- **X1**, **C1**
- **L**, **A1**

Cycle run:

1   Activate the C axis and position to **Spindle angle C** at rapid traverse (only in **Teach-in** submode)
2   Calculate the proportioning of cuts
3   Approach at **Approach feed FZ**
4   Machine to the **end point of slot**
5   Approach at **Approach feed FZ**
6   Machine to the **starting point of slot**
7   Repeat 3 to 6 until the milling depth is reached
8   Position to the **Start point X** and deactivate the C axis
9   Move to the **Tool change point** according to the setting in **G14**

Axial figure

▶   Select **Milling**

▶   Select **Axial figure**

Depending on the parameters, the cycle mills one of the following contours or roughs/finishes a pocket on the face:

- Rectangle (**Q** = 4, **L** <> **B**)
- Square (**Q** = 4, **L** = **B**)
- Circle (**Q** = 0, **RE** > 0, **L** and **B**: no input)
- Triangle or polygon (**Q** = 3 or **Q** > 4, **L** <> 0)

Cycle parameters:

- **X**, **Z**: **Start point**
- **C**: **Spindle angle** – C-axis position (default: current spindle angle)
- **X1**: **Dia. of figure center**
- **C1**: **Angle of figure center** (default: **Spindle angle C**)
- **Q**: **Number edges** (default: 0)
    - **Q** = 0: Circle
    - **Q** = 4: Rectangle, square
    - **Q** = 3: Triangle
    - **Q** > 4: Polygon
- **L**: **Edge length**
    - Rectangle: Rectangle length
    - Square, polygon: Edge length
    - Polygon: **L** < 0 inscribed circle diameter
    - Circle: No input
- **B**: **Width of rectangle**
    - Rectangle: Rectangle width
    - Square: **L** = **B**
    - Polygon, circle: No input
- **RE**: **Rounding radius** (default: 0)
    - Rectangle, square, polygon: Rounding radius
    - Circle: Circle radius
- **A**: **Angle to X axis** (default: 0°)
    - Rectangle, square, polygon: Position of figure
    - Circle: No input
- **Z1**: **Millg. top edge** (default: **Starting point Z**)
- **P2**: **Milling depth**
- **T**: **Tool number** – turret pocket number

- **G14**: **Tool change point**
  Further information: "Tool change point G14", Page 203
- **ID**: **ID no.**
- **S**: **Cutting speed** or **Constant speed**
- **F**: **Feed per revolution**
- **I**: **Contour-parallel oversize**
- **K**: **Infeed-direction oversize**
- **P**: **Feed depth** (default: total depth in one infeed)
- **FZ**: **Approach feed** (default: active feed rate)
- **E**: **Reduced feed** for circular elements (default: active feed rate)
- **EW**: **Plunging angle**
- **O**: **Roughing/Finish** − Only for pocket milling
  - **0: Roughing**
  - **1: Finishing**
- **H**: **Mill cutting direction**
  - **0: Up-cut**
  - **1: Climb**
- **U**: **Overlap factor** − defines the overlap of milling paths (default: 0.5) (range: 0 to 0.99)
  Overlap = **U** * milling diameter
  - **U** = 0 or no input: Contour milling
  - **U** > 0: Pocket milling − Minimum overlap of milling paths = **U** * milling diameter
- **QK**: **TYPE OF INFEED** (default: 0)
  - 0: Linear
  - 1: Reciprocating
  - 2: Helical
- **JK**: **Contour milling** − Input is evaluated only for contour milling
  - **0: On the contour**
  - **1: Within the contour**
  - **2: Outside the contour**
- **JT**: **Pocket milling** − Input is evaluated only for pocket milling
  - **0: From the inside out**
  - **1: From the outside in**
- **R**: **Apprch angle** (default: 0)
  - **R** = 0: Contour element is approached directly; infeed to starting point above the milling plane—then vertical plunge
  - **R** > 0: Tool moves on approaching/departing arc that connects tangentially to the contour element
  - **R** < 0 for inside corners: Tool moves on approaching/departing arc that connects tangentially to the contour element
  - **R** < 0 for outside corners: Length of linear approaching/departing element; contour element is approached/departed tangentially

- **RB**: **Return plane**
- **SCI**: **Safety clearance** in the working plane
- **SCK**: **Safety clearance** in infeed direction
  Further information: "Safety clearances SCI and SCK", Page 203
- **MT**: **M after T**: **M** function that is executed after the **T** tool call
- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step
- **MFE**: **M at end**: **M** function that is executed at the end of the machining step
- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
  - Main drive
  - Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)

> **ℹ** Type of machining for technology database access: **Milling**

> **ℹ** Notes on parameters and functions:
> - **Machining of contours or pockets:** Defined with **Overlap factor U**
> - **Milling direction:** Depends on the definition of **Mill cutting direction H** and the direction of rotation of the milling cutter
>   **Further information:** "Milling direction for contour milling", Page 409
> - **Milling cutter radius compensation:** Effective (except for contour milling with **J**=0)
> - **Approach and departure:** For closed contours, the starting point of the first element (in rectangles, the longer element) is the point of approach and departure. The tool approaches directly or on an arc according to **Approach radius R**
> - **Contour milling JK** defines whether the milling cutter will machine on the contour (center of milling cutter on the contour) or inside/outside of the contour
> - **Pocket milling—roughing (O=0):** Use **JT** to define whether a pocket is machined from the inside toward the outside, or vice versa
> - **Pocket milling—finishing (O=1):** First, the edge of the pocket is machined; then the pocket floor is machined. With **JT** you define whether a pocket floor is to be finished from the inside towards the outside, or vice versa

Cycle run:

All versions:

1   Activate the C axis and position to **Spindle angle C** at rapid traverse (only in **Teach-in** submode)

2   Calculate the proportioning of cuts (infeeds to the milling planes, infeeds in the milling planes)

   Contour milling:

   3   Depending on the **Apprch angle R**, approach the workpiece and plunge to the first milling plane

   4   Mill the first plane

   5   Plunge to the next milling plane

   6   Repeat 4 to 5 until the milling depth is reached

   Pocket milling—roughing:

   3   Move to the **Safety clearance** and plunge to the first milling plane

   4   Depending on **Pocket milling JT**, machine the milling plane either from the inside towards the outside, or vice versa

   5   Plunge to the next milling plane

   6   Repeat 4 to 5 until the milling depth is reached

   Pocket milling—finishing:

   3   Depending on the **Apprch angle R**, approach the workpiece and plunge to the first milling plane

   4   Finish-machine the edge of the pocket—one working plane after the other

   5   Depending on **Pocket milling JT**, finish the pocket floor either from the inside towards the outside, or vice versa

   6   Finish-machine the pocket at the programmed feed rate

All versions:

7   Position to the **Start point Z** and deactivate the C axis

8   Move to the **Tool change point** according to the setting in **G14**
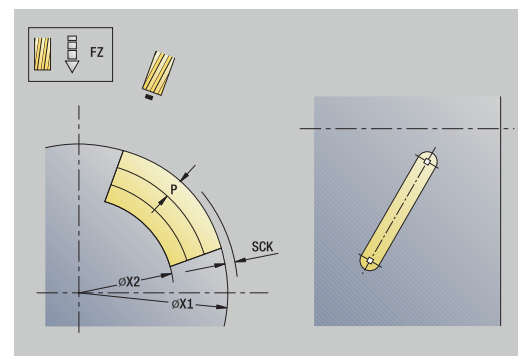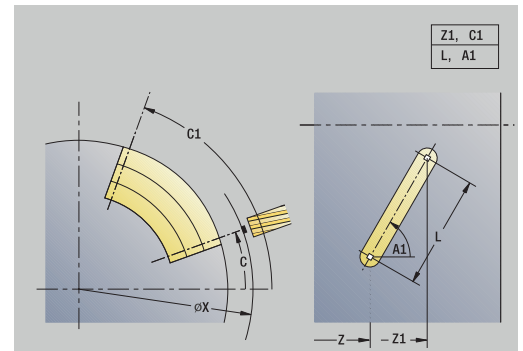
Radial figure

▶ Select **Milling**

▶ Select **Radial figure**

Depending on the parameters, the cycle mills one of the following contours or roughs/finishes a pocket on the lateral surface:

- Rectangle (**Q**=4, **L**<>**B**)
- Square (**Q**=4, **L**=**B**)
- Circle (**Q**=0, **RE**>0, **L** and **B**: no input)
- Triangle or polygon (**Q**=3 or **Q**>4, **L**<>0)

Cycle parameters:

- **X**, **Z**: **Start point**
- **C**: **Spindle angle** – C-axis position (default: current spindle angle)
- **Z1**: **Figure center**
- **C1**: **Angle of figure center** (default: **Spindle angle C**)
- **Q**: **Number edges** (default: 0)
  - **Q** = 0: Circle
  - **Q** = 4: Rectangle, square
  - **Q** = 3: Triangle
  - **Q** > 4: Polygon
- **L**: **Edge length**
  - Rectangle: Rectangle length
  - Square, polygon: Edge length
  - Polygon: **L** < 0 inscribed circle diameter
  - Circle: No input
- **B**: **Width of rectangle**
  - Rectangle: Rectangle width
  - Square: **L** = **B**
  - Polygon, circle: No input
- **RE**: **Rounding radius** (default: 0)
  - Rectangle, square, polygon: Rounding radius
  - Circle: Circle radius
- **A**: **Angle to Z axis** (default: 0°)
  - Rectangle, square, polygon: Position of figure
  - Circle: No input
- **X1**: **Upper edge of milling** (diameter value; default: **Starting point X**)
- **P2**: **Milling depth**
- **T**: **Tool number** – turret pocket number
- **G14**: **Tool change point**
  Further information: "Tool change point G14", Page 203
- **ID**: **ID no.**
- **S**: **Cutting speed** or **Constant speed**
- **F**: **Feed per revolution**
- **I**: **Infeed-direction oversize**
- **K**: **Contour-parallel oversize**

- **P**: **Feed depth** (default: total depth in one infeed)
- **FZ**: **Approach feed** (default: active feed rate)
- **E**: **Reduced feed** for circular elements (default: active feed rate)
- **EW**: **Plunging angle**
- **O**: **Roughing/Finish** – Only for pocket milling
    - **0: Roughing**
    - **1: Finishing**
- **H**: **Mill cutting direction**
    - **0: Up-cut**
    - **1: Climb**
- **U**: **Overlap factor** – defines the overlap of milling paths (default: 0.5) (range: 0 to 0.99)
  Overlap = **U** * milling diameter
    - **U** = 0 or no input: Contour milling
    - **U** > 0: Pocket milling – Minimum overlap of milling paths = **U** * milling diameter
- **QK**: **TYPE OF INFEED** (default: 0)
    - 0: Linear
    - 1: Reciprocating
    - 2: Helical
- **JK**: **Contour milling** – Input is evaluated only for contour milling
    - **0: On the contour**
    - **1: Within the contour**
    - **2: Outside the contour**
- **JT**: **Pocket milling** – Input is evaluated only for pocket milling
    - **0: From the inside out**
    - **1: From the outside in**
- **R**: **Apprch angle** (default: 0)
    - **R** = 0: Contour element is approached directly; infeed to starting point above the milling plane—then vertical plunge
    - **R** > 0: Tool moves on approaching/departing arc that connects tangentially to the contour element
    - **R** < 0 for inside corners: Tool moves on approaching/ departing arc that connects tangentially to the contour element
    - **R** < 0 for outside corners: Length of linear approaching/ departing element; contour element is approached/departed tangentially
- **RB**: **Return plane**
- **SCI**: **Safety clearance** in the working plane
- **SCK**: **Safety clearance** in infeed direction
  Further information: "Safety clearances SCI and SCK", Page 203
- **MT**: **M after T**: **M** function that is executed after the **T** tool call
- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step
- **MFE**: **M at end**: **M** function that is executed at the end of the machining step
- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)

- Main drive
- Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)

> **ⓘ** Type of machining for technology database access: **Milling**

> **ⓘ** Notes on parameters and functions:
> - **Machining of contours or pockets:** Defined with **Overlap factor U**
> - **Milling direction:** Depends on the definition of **Mill cutting direction H** and the direction of rotation of the milling cutter
>   **Further information:** "Milling direction for contour milling", Page 409
> - **Milling cutter radius compensation:** Effective (except for contour milling with **J**=0)
> - **Approach and departure:** For closed contours, the starting point of the first element (in rectangles, the longer element) is the point of approach and departure. The tool approaches directly or on an arc according to **Approach radius R**
> - **Contour milling JK** defines whether the milling cutter will machine on the contour (center of milling cutter on the contour) or inside/outside of the contour
> - **Pocket milling—roughing (O=0):** Use **JT** to define whether a pocket is machined from the inside toward the outside, or vice versa
> - **Pocket milling—finishing (O=1):** First, the edge of the pocket is machined; then the pocket floor is machined. With **JT** you define whether a pocket floor is to be finished from the inside towards the outside, or vice versa

Cycle run:

All versions:

1　Activate the C axis and position to **Spindle angle C** at rapid traverse (only in **Teach-in** submode)

2　Calculate the proportioning of cuts (infeeds to the milling planes, infeeds in the milling planes)

Contour milling:

3　Depending on the **Apprch angle R**, approach the workpiece and plunge to the first milling plane

4　Mill the first plane

5　Plunge to the next milling plane

6　Repeat 4 to 5 until the milling depth is reached

Pocket milling—roughing:

3　Move to the **Safety clearance** and plunge to the first milling plane

4　Depending on **Pocket milling JT**, machine the milling plane either from the inside towards the outside, or vice versa

5　Plunge to the next milling plane

6　Repeat 4 to 5 until the milling depth is reached

Pocket milling—finishing:

3　Depending on the **Apprch angle R**, approach the workpiece and plunge to the first milling plane

4　Finish-machine the edge of the pocket—one working plane after the other

5　Depending on **Pocket milling JT**, finish the pocket floor either from the inside towards the outside, or vice versa

6　Finish-machine the pocket at the programmed feed rate

All versions:

7　Position to the **Start point Z** and deactivate the C axis

8　Move to the **Tool change point** according to the setting in **G14**

ICP contour axial

▶ Select **Milling**

▶ Select **ICP contour axial**

Depending on the parameters, the cycle mills one of the following contours or roughs/finishes a pocket on the face.
Cycle parameters:

- **X**, **Z**: **Start point**
- **C**: **Spindle angle** – C-axis position
- **Z1**: **Millg. top edge** (default: **Starting point Z**)
- **P2**: **Milling depth**
- **I**: **Contour-parallel oversize**
- **K**: **Infeed-direction oversize**
- **P**: **Feed depth** (default: total depth in one infeed)
- **FZ**: **Approach feed** (default: active feed rate)
- **E**: **Reduced feed** for circular elements (default: active feed rate)
- **EW**: **Plunging angle**
- **FK**: **ICP contour number**
- **T**: **Tool number** – turret pocket number
- **G14**: **Tool change point**
  Further information: "Tool change point G14", Page 203
- **ID**: **ID no.**
- **S**: **Cutting speed** or **Constant speed**
- **F**: **Feed per revolution**
- **O**: **Roughing/Finish** – Only for pocket milling
  - **0: Roughing**
  - **1: Finishing**
- **H**: **Mill cutting direction**
  - **0: Up-cut**
  - **1: Climb**
- **U**: **Overlap factor** – defines the overlap of milling paths (default: 0.5) (range: 0 to 0.99)
  Overlap = **U** * milling diameter
  - **U** = 0 or no input: Contour milling
  - **U** > 0: Pocket milling – Minimum overlap of milling paths = **U** * milling diameter
- **QK**: **TYPE OF INFEED** (default: 0)
  - 0: Linear
  - 1: Reciprocating
  - 2: Helical

- **JK**: **Contour milling** – Input is evaluated only for contour milling
  - **0: On the contour**
  - **1: Within the contour**
  - **2: Outside the contour**
- **JT**: **Pocket milling** – Input is evaluated only for pocket milling
  - **0: From the inside out**
  - **1: From the outside in**
- **R**: **Apprch angle** (default: 0)
  - **R** = 0: Contour element is approached directly; infeed to starting point above the milling plane—then vertical plunge
  - **R** > 0: Tool moves on approaching/departing arc that connects tangentially to the contour element
  - **R** < 0 for inside corners: Tool moves on approaching/departing arc that connects tangentially to the contour element
  - **R** < 0 for outside corners: Length of linear approaching/departing element; contour element is approached/departed tangentially
- **RB**: **Return plane**
- **SCI**: **Safety clearance** in the working plane
- **SCK**: **Safety clearance** in infeed direction
  Further information: "Safety clearances SCI and SCK", Page 203
- **BG**: **Cham. width** for deburring
- **JG**: **Preparation diameter**
- **MT**: **M after T**: **M** function that is executed after the **T** tool call
- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step
- **MFE**: **M at end**: **M** function that is executed at the end of the machining step
- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
  - Main drive
  - Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)

> **ⓘ** Type of machining for technology database access: **Milling**

ⓘ Notes on parameters and functions:
- **Machining of contours or pockets:** Defined with **Overlap factor U**
- **Milling direction:** Depends on the definition of **Mill cutting direction H** and the direction of rotation of the milling cutter
  Further information: "Milling direction for contour milling", Page 409
- **Milling cutter radius compensation:** Effective (except for contour milling with **J**=0)
- **Approach and departure:** For closed contours, the starting point of the first element (in rectangles, the longer element) is the point of approach and departure. The tool approaches directly or on an arc according to **Approach radius R**
- **Contour milling JK** defines whether the milling cutter will machine on the contour (center of milling cutter on the contour) or inside/outside of the contour
- **Pocket milling—roughing (O=0):** Use **JT** to define whether a pocket is machined from the inside toward the outside, or vice versa
- **Pocket milling—finishing (O=1):** First, the edge of the pocket is machined; then the pocket floor is machined. With **JT** you define whether a pocket floor is to be finished from the inside towards the outside, or vice versa

Cycle run:

All versions:

1 Activate the C axis and position to **Spindle angle C** at rapid traverse (only in **Teach-in** submode)

2 Calculate the proportioning of cuts (infeeds to the milling planes, infeeds in the milling planes)

Contour milling:

3 Depending on the **Apprch angle R**, approach the workpiece and plunge to the first milling plane

4 Mill the first plane

5 Plunge to the next milling plane

6 Repeat 4 to 5 until the milling depth is reached

Pocket milling—roughing:

3 Move to the **Safety clearance** and plunge to the first milling plane

4 Depending on **Pocket milling JT**, machine the milling plane either from the inside towards the outside, or vice versa

5 Plunge to the next milling plane

6 Repeat 4 to 5 until the milling depth is reached

Pocket milling—finishing:

3 Depending on the **Apprch angle R**, approach the workpiece and plunge to the first milling plane

4 Finish-machine the edge of the pocket—one working plane after the other

5 Depending on **Pocket milling JT**, finish the pocket floor either from the inside towards the outside, or vice versa

6 Finish-machine the pocket at the programmed feed rate

All versions:

7 Position to the **Start point Z** and deactivate the C axis

8 Move to the **Tool change point** according to the setting in **G14**

ICP contour radial

▶ Select **Milling**

▶ Select **ICP contour radial**

Depending on the parameters, the cycle mills one of the following contours or roughs/finishes a pocket on the lateral surface.

Cycle parameters:

- **X**, **Z**: **Start point**
- **C**: **Spindle angle** – C-axis position
- **X1**: **Upper edge of milling** (diameter value; default: **Starting point X**)
- **P2**: **Milling depth**
- **I**: **Infeed-direction oversize**
- **K**: **Contour-parallel oversize**
- **P**: **Feed depth** (default: total depth in one infeed)
- **FZ**: **Approach feed** (default: active feed rate)
- **E**: **Reduced feed** for circular elements (default: active feed rate)
- **EW**: **Plunging angle**
- **FK**: **ICP contour number**
- **T**: **Tool number** – turret pocket number
- **G14**: **Tool change point**
  Further information: "Tool change point G14", Page 203
- **ID**: **ID no.**
- **S**: **Cutting speed** or **Constant speed**
- **F**: **Feed per revolution**
- **O**: **Roughing/Finish** – Only for pocket milling
    - **0: Roughing**
    - **1: Finishing**
- **H**: **Mill cutting direction**
    - **0: Up-cut**
    - **1: Climb**
- **U**: **Overlap factor** – defines the overlap of milling paths (default: 0.5) (range: 0 to 0.99)
  Overlap = **U** * milling diameter
    - **U** = 0 or no input: Contour milling
    - **U** > 0: Pocket milling – Minimum overlap of milling paths = **U** * milling diameter
- **QK**: **TYPE OF INFEED** (default: 0)
    - 0: Linear
    - 1: Reciprocating
    - 2: Helical

- **JK**: **Contour milling** – Input is evaluated only for contour milling
    - **0: On the contour**
    - **1: Within the contour**
    - **2: Outside the contour**
- **JT**: **Pocket milling** – Input is evaluated only for pocket milling
    - **0: From the inside out**
    - **1: From the outside in**
- **R**: **Apprch angle** (default: 0)
    - **R** = 0: Contour element is approached directly; infeed to starting point above the milling plane—then vertical plunge
    - **R** > 0: Tool moves on approaching/departing arc that connects tangentially to the contour element
    - **R** < 0 for inside corners: Tool moves on approaching/departing arc that connects tangentially to the contour element
    - **R** < 0 for outside corners: Length of linear approaching/departing element; contour element is approached/departed tangentially
- **RB**: **Return plane**
- **SCI**: **Safety clearance** in the working plane
- **SCK**: **Safety clearance** in infeed direction
  Further information: "Safety clearances SCI and SCK", Page 203
- **BG**: **Cham. width** for deburring
- **JG**: **Preparation diameter**
- **MT**: **M after T**: **M** function that is executed after the **T** tool call
- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step
- **MFE**: **M at end**: **M** function that is executed at the end of the machining step
- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
    - Main drive
    - Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)

> **ℹ** Type of machining for technology database access: **Milling**

ℹ️ Notes on parameters and functions:
- **Machining of contours or pockets:** Defined with **Overlap factor U**
- **Milling direction:** Depends on the definition of **Mill cutting direction H** and the direction of rotation of the milling cutter
  Further information: "Milling direction for contour milling", Page 409
- **Milling cutter radius compensation:** Effective (except for contour milling with **J**=0)
- **Approach and departure:** For closed contours, the starting point of the first element (in rectangles, the longer element) is the point of approach and departure. The tool approaches directly or on an arc according to **Approach radius R**
- **Contour milling JK** defines whether the milling cutter will machine on the contour (center of milling cutter on the contour) or inside/outside of the contour
- **Pocket milling—roughing (O=0):** Use **JT** to define whether a pocket is machined from the inside toward the outside, or vice versa
- **Pocket milling—finishing (O=1):** First, the edge of the pocket is machined; then the pocket floor is machined. With **JT** you define whether a pocket floor is to be finished from the inside towards the outside, or vice versa

Cycle run:

All versions:

1 Activate the C axis and position to **Spindle angle C** at rapid traverse (only in **Teach-in** submode)

2 Calculate the proportioning of cuts (infeeds to the milling planes, infeeds in the milling planes)

Contour milling:

3 Depending on the **Apprch angle R**, approach the workpiece and plunge to the first milling plane

4 Mill the first plane

5 Plunge to the next milling plane

6 Repeat 4 to 5 until the milling depth is reached

Pocket milling—roughing:

3 Move to the **Safety clearance** and plunge to the first milling plane

4 Depending on **Pocket milling JT**, machine the milling plane either from the inside towards the outside, or vice versa

5 Plunge to the next milling plane

6 Repeat 4 to 5 until the milling depth is reached

Pocket milling—finishing:

3 Depending on the **Apprch angle R**, approach the workpiece and plunge to the first milling plane

4 Finish-machine the edge of the pocket—one working plane after the other

5 Depending on **Pocket milling JT**, finish the pocket floor either from the inside towards the outside, or vice versa

6 Finish-machine the pocket at the programmed feed rate

All versions:

7 Position to the **Start point Z** and deactivate the C axis

8 Move to the **Tool change point** according to the setting in **G14**

Face milling

▶ Select **Milling**

▶ Select **Face milling**

Depending on the parameters, the cycle mills the following contours on the face:

- One or two surfaces (**Q** = 1 or **Q** = 2, **B** > 0)
- Rectangle (**Q** = 4, **L** <> **B**)
- Square (**Q** = 4, **L** = **B**)
- Triangle or polygon (**Q** = 3 or **Q** > 4, **L** <> 0)
- Circle (**Q** = 0, **RE** > 0, **L** and **B**: no input)
- ICP contour (**Expanded** soft key, entry in **FK**)

Cycle parameters:

- **X**, **Z**: **Start point**
- **C**: **Spindle angle** − C-axis position (default: current spindle angle)
- **X1**: **Dia. of figure center**
- **C1**: **Angle of figure center** (default: **Spindle angle C**)
- **Z1**: **Millg. top edge** (default: **Starting point Z**)
- **Z2**: **Milling floor**
- **FK**: **ICP contour number**
- **Q**: **Number edges**
  - **Q** = 0: Circle
  - **Q** = 1: One surface
  - **Q** = 2: Two surfaces offset by 180°
  - **Q** = 3: Triangle
  - **Q** = 4: Rectangle, square
  - **Q** > 4: Polygon
- **L**: **Edge length**
  - Rectangle: Rectangle length
  - Square, polygon: Edge length
  - Polygon: **L** < 0 inscribed circle diameter
  - Circle: No input
- **B**: **Width/Width across flats**
  - For **Q** = 1, **Q** = 2: Remaining thickness (remaining material)
  - Rectangle: Rectangle width
  - Square, polygon (**Q** >= 4): Width across flats (use only for even number of surfaces; program "B" as an alternative to **L**)
  - Circle: No input
- **RE**: **Rounding radius** (default: 0)
  - Polygon (**Q** > 2): Rounding radius
  - Circle (**Q** = 0): Radius of circle
- **A**: **Angle to X axis** (default: 0°)
  - Polygon (**Q** > 2): Position of figure
  - Circle: No input
- **T**: **Tool number** − turret pocket number
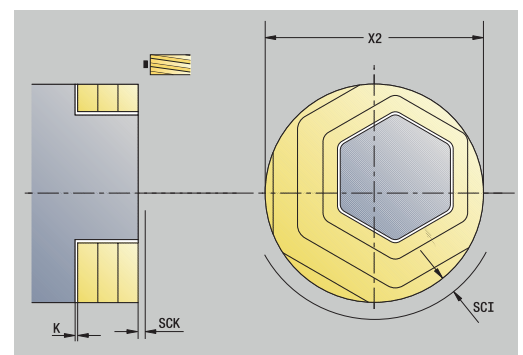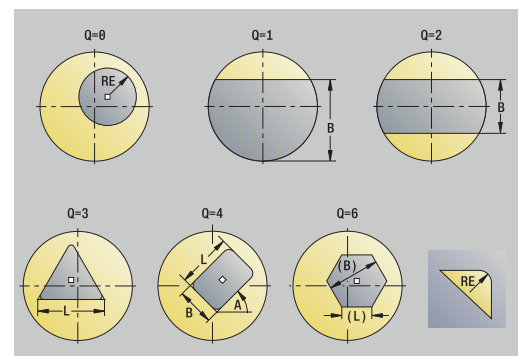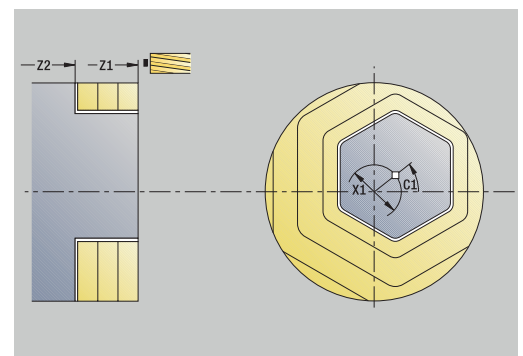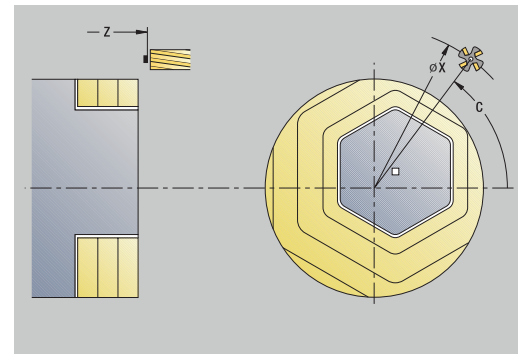
- **G14**: **Tool change point**
  Further information: "Tool change point G14", Page 203
- **ID**: **ID no.**
- **S**: **Cutting speed** or **Constant speed**
- **F**: **Feed per revolution**
- **I**: **Contour-parallel oversize**
- **K**: **Infeed-direction oversize**
- **X2**: **Limit diameter**
- **P**: **Feed depth** (default: total depth in one infeed)
- **FZ**: **Approach feed** (default: active feed rate)
- **E**: **Reduced feed** for circular elements (default: active feed rate)
- **U**: **Overlap factor** – defines the overlap of milling paths (default: 0.5) (range: 0 to 0.99)
  Overlap = **U** * milling diameter
- **O**: **Roughing/Finish**
  - **0: Roughing**
  - **1: Finishing**
- **H**: **Mill cutting direction**
  - **0: Up-cut**
  - **1: Climb**
- **J**: **Mill direction**
- **SCI**: **Safety clearance** in the working plane
- **SCK**: **Safety clearance** in infeed direction
  Further information: "Safety clearances SCI and SCK", Page 203
- **MT**: **M after T**: **M** function that is executed after the **T** tool call
- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step
- **MFE**: **M at end**: **M** function that is executed at the end of the machining step
- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
  - Main drive
  - Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)

> **ⓘ** Type of machining for technology database access: **Milling**

Cycle run:

All versions:

1 Activate the C axis and position to **Spindle angle C** at rapid traverse (only in **Teach-in** submode)

2 Calculate the proportioning of cuts (infeeds to the milling planes, infeeds in the milling planes)

3 Move to the **Safety clearance** and plunge to the first milling plane

Roughing:

4 Machine the milling plane, taking **Mill direction J** (unidirectional or bidirectional) into account

5 Plunge to the next milling plane

6 Repeat 4 to 5 until the milling depth is reached

Finishing:

4 Finish-machine the edge of the island—one working plane after the other

5 Finish-machine the floor from the outside towards the inside

All versions:

7 Position to the **Start point Z** and deactivate the C axis

8 Move to the **Tool change point** according to the setting in **G14**

Hel. slot milling, radial

▶ Select **Milling**

▶ Select **Hel. slot milling, radial**

The cycle mills a helical slot from the **Start point thread** to the **End point thread**. The **Start. angle** defines the starting position for the slot. The slot width equals the diameter of the milling cutter.

Cycle parameters:

- **X**, **Z**: **Start point**
- **C**: **Spindle angle** – C-axis position
- **X1**: **Thread diameter**
- **C1**: **Start. angle**
- **Z1**: **Start point thread**
- **Z2**: **End point thread**
- **F1**: **Thread pitch**
  - **F1** positive: Right-hand thread
  - **F1** negative: Left-hand thread
- **U**: **Thread depth**
- **I**: **Maximum infeed** – The infeed movements are reduced down to >= 0.5 mm according to the following calculation. Each following infeed movement will then be 0.5 mm
  - Infeed 1: **I**
  - Infeed n: **I** * (1 − (n − 1) * **E**)
- **E**: **Cutting depth reduction**
- **P**: **Run-in lgth** – Ramp at the beginning of the slot
- **K**: **Thread runout length** – Ramp at the end of the slot
- **T**: **Tool number** – turret pocket number
- **G14**: **Tool change point**
  Further information: "Tool change point G14", Page 203
- **ID**: **ID no.**
- **S**: **Cutting speed** or **Constant speed**
- **F**: **Feed per revolution**
- **D**: **No.gears**
- **SCK**: **Safety clearance** in infeed direction
  Further information: "Safety clearances SCI and SCK", Page 203
- **MT**: **M after T**: **M** function that is executed after the **T** tool call
- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step
- **MFE**: **M at end**: **M** function that is executed at the end of the machining step
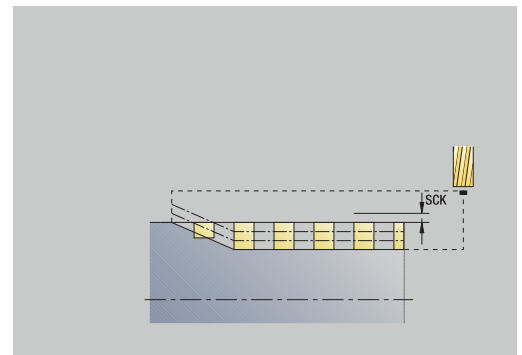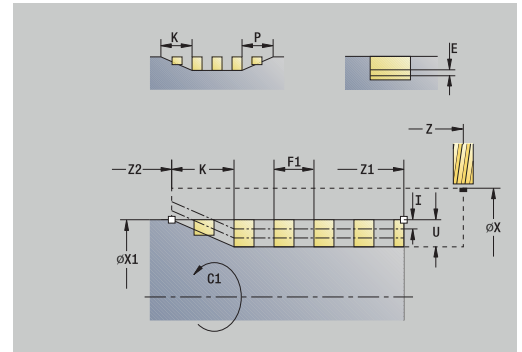
- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
  - Main drive
  - Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)

> 🛈 Type of machining for technology database access: **Milling**

Cycle run:

1. Activate the C axis and position to **Spindle angle C** at rapid traverse (only in **Teach-in** submode)
2. Calculate the current infeed
3. Position the tool for the first pass
4. Machine up to the **End point thread Z2** at the programmed feed rate, taking the ramps at the beginning and end of the slot into account
5. Return on a paraxial path and approach for the next pass
6. Repeat 4 to 5 until the slot depth is reached
7. Move to the **Tool change point** according to the setting in **G14**

Milling direction for contour milling

Cycle type	Cutting direction	Direction of tool rotation	MCRC	Execution
Inside (**JK**=1)	Up-cut milling (**H**=0)	Mx03	Right	
Inside	Up-cut milling (**H**=0)	Mx04	Left	
Inside	Climb milling (**H**=1)	Mx03	Left	
Inside	Climb milling (**H**=1)	Mx04	Right	
Outside (**JK**=2)	Up-cut milling (**H**=0)	Mx03	Right	
Outside	Up-cut milling (**H**=0)	Mx04	Left	
Outside	Climb milling (**H**=1)	Mx03	Left	
Outside	Climb milling (**H**=1)	Mx04	Right	
Right (**JK**=2)	For open contours without function. Machining in direction of contour definition	Without effect	Right	
Left (**JK**=1)	For open contours without function. Machining in direction of contour definition	Without effect	Left	

Cycle type	Cutting direction	Direction of tool rotation	MCRC	Execution
Roughing Finishing	Up-cut milling (**H**=0)	From the inside toward the outside (JT=0)	Mx03	
Roughing Finishing	Up-cut milling (**H**=0)	From the inside toward the outside (**JT**=0)	Mx04	
Roughing	Climb milling (**H**=0)	From the outside toward the inside (**JT**=1)	Mx03	
Roughing	Up-cut milling (**H**=0)	From the outside toward the inside (**JT**=1)	Mx04	
Roughing Finishing	Climb milling (**H**=1)	From the inside toward the outside (**JT**=0)	Mx03	
Roughing Finishing	Climb milling (**H**=1)	From the inside toward the outside (**JT**=0)	Mx04	
Roughing	Climb milling (**H**=1)	From the outside toward the inside (**JT**=1)	Mx03	
Roughing	Up-cut milling (**H**=1)	From the outside toward the inside (**JT**=1)	Mx04	

Examples of milling cycles

Milling on the face

In this example, a pocket is milled. The milling example illustrates the complete machining process on the face, including contour definition.

The machining process is performed with the cycle **ICP contour axial**. To describe a contour, define the basic contour first. Then superimpose the rounding arcs.

Tool data (milling cutter)

- **TO** = 8 – Tool orientation
- **I** = 8 – Milling cutter diameter
- **K** = 4 – Number of teeth
- **TF** = 0.025 – Feed per tooth
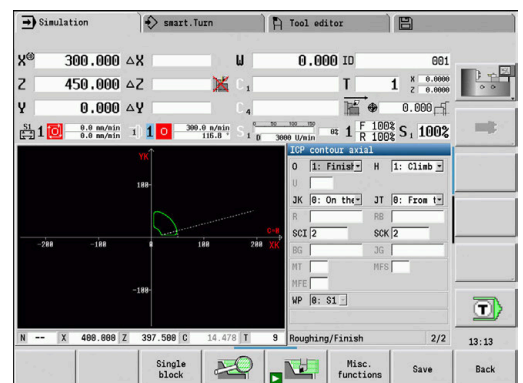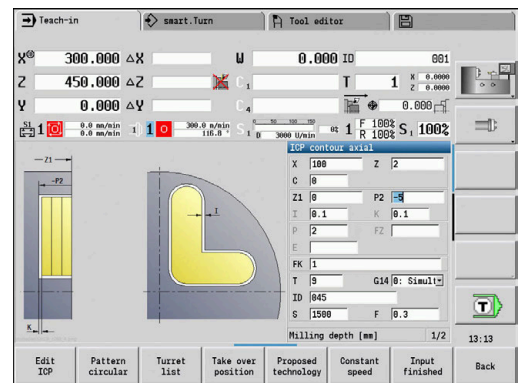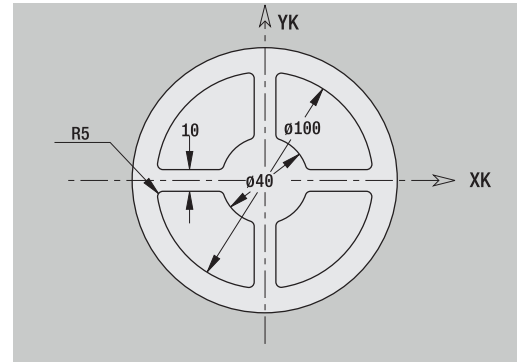
Engraving, axial

Engraving, axial

▶ Select **Milling**

▶ Select **Engraving**

▶ Select **Engraving, axial**

The **Engraving, axial** cycle engraves character strings in linear or polar layout on the face of the workpiece.

Character set and more information:

Further information: "Engraving, axial and radial", Page 415

Define the **Start point** of the character string in the cycle. If you do not define a **Start point**, the cycle starts at the current tool position.

You can also use multiple calls to engrave a logotype. For this purpose, specify the **Start point** with the first call. All other calls are programmed without a **Start point**.

Cycle parameters:

- **X**: **Starting point** – Pre-positioning the tool (diameter value)
- **Z**: **Starting point** – Pre-positioning the tool
- **C**: **Spindle angle** – Pre-positioning the workpiece spindle
- **TX**: **Text** to be engraved
- **NF**: **Char. no.** – ASCII code of the character to be engraved
- **Z2**: **Final point** – Z position, infeed depth during engraving
- **X1**: **Start point** of first character (in polar coordinates)
- **C1**: **Start. angle** (polar) of first character
- **XK**: **Start point** of first character (in Cartesian coordinates)
- **YK**: **Start point** of first character (in Cartesian coordinates)
- **H**: **Font height**
- **E**: **Distance factor**
  The spacing between the characters is calculated according to the following formula: **H** / 6 * **E**
- **T**: **Tool number** – turret pocket number
- **G14**: **Tool change point**
  Further information: "Tool change point G14", Page 203
- **ID**: **ID no.**
- **S**: **Cutting speed** or **Constant speed**
- **F**: **Feed per revolution**
- **W**: **Inclinat. ang.** of the character string
- **FZ**: **Plunging feed rate factor** (plunging feed rate = current feed rate * **FZ**)
- **V**: **Execution (linear/polar)**
- **D**: **Reference diameter**
- **RB**: **Return plane** – Z position to which the tool retracts for positioning
- **SCK**: **Safety clearance**
  Further information: "Safety clearances SCI and SCK", Page 203
- **MT**: **M after T**: **M** function that is executed after the **T** tool call

- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step
- **MFE**: **M at end**: **M** function that is executed at the end of the machining step
- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
  - Main drive
  - Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)

> **i** The engraving cycles are not available in **Machine** operating mode.

Cycle run:
1 Activate the C axis and position to the **Spindle angle C**, **Start point X** and **Z** at rapid traverse
2 Position to the **Start point**, if defined
3 Infeed at **Plunging feed rate factor FZ**
4 Engrave at the programmed feed rate
5 Position the tool to the **Return plane RB** or, if **RB** is not defined, to the **Start point Z**
6 Position the tool to the next character
7 Repeat steps 3 to 6 until all characters are engraved
8 Position to the **Start point X**, **Z** and deactivate the C axis
9 Move to the **Tool change point** according to the setting in **G14**

Engraving, radial

▶ Select **Milling**

▶ Select **Engraving**

▶ Select **Engraving, radial**

The **Engraving, radial** cycle engraves character strings in linear arrangement on the lateral surface of the workpiece.

Character set and more information:

Further information: "Engraving, axial and radial", Page 415

Define the **Start point** of the character string in the cycle. If you do not define a **Start point**, the cycle starts at the current tool position.

You can also use multiple calls to engrave a logotype. For this purpose, specify the **Start point** with the first call. All other calls are programmed without a **Start point**.

Cycle parameters:

- **X**: **Starting point** – Pre-positioning the tool (diameter value)
- **Z**: **Starting point** – Pre-positioning the tool
- **C**: **Spindle angle** – Pre-positioning the workpiece spindle
- **TX**: **Text** to be engraved
- **NF**: **Char. no.** – ASCII code of the character to be engraved
- **X2**: **Final point** – X position, infeed depth during engraving (diameter value)
- **Z1**: **Start point** of first character
- **C1**: **Start. angle** of first character
- **CY**: **Start point** of first character
- **D**: **Reference diameter**
- **H**: **Font height**
- **E**: **Distance factor**
  The spacing between the characters is calculated according to the following formula: **H** / 6 * **E**
- **T**: **Tool number** – turret pocket number
- **G14**: **Tool change point**
  Further information: "Tool change point G14", Page 203
- **ID**: **ID no.**
- **S**: **Cutting speed** or **Constant speed**
- **F**: **Feed per revolution**
- **W**: **Inclinat. ang.** of the character string
- **FZ**: **Plunging feed rate factor** (plunging feed rate = current feed rate * **FZ**)
- **RB**: **Return plane** – X position to which the tool retracts for positioning
- **SCK**: **Safety clearance**
  Further information: "Safety clearances SCI and SCK", Page 203
- **MT**: **M after T**: **M** function that is executed after the **T** tool call
- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step

- **MFE**: **M at end**: **M** function that is executed at the end of the machining step
- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
  - Main drive
  - Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
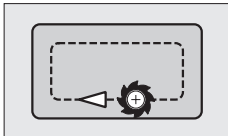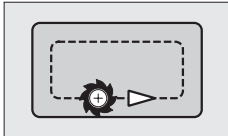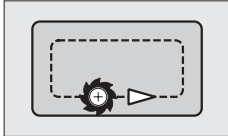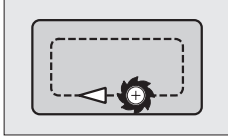- **DF**: **Miscellaneous function** (machine-dependent)

> **i** The engraving cycles are not available in **Machine** operating mode.

Cycle run:
1. Activate the C axis and position to the **Spindle angle C**, **Start point X** and **Z** at rapid traverse
2. Position to the **Start point**, if defined
3. Infeed at **Plunging feed rate factor FZ**
4. Engrave at the programmed feed rate
5. Position the tool to the **Return plane RB** or, if **RB** is not defined, to the **Start point X**
6. Position the tool to the next character
7. Repeat steps 3 to 5 until all characters are engraved
8. Position to the **Start point X**, **Z** and deactivate the C axis
9. Move to the **Tool change point** according to the setting in **G14**

Engraving, axial and radial

Engraving, axial and radial

The control can realize the characters listed in the following table. The text to be engraved is entered as a character string. Diacritics and special characters that you cannot enter in the editor can be defined, character by character, in **NF**. If text is defined in **ID** and a character is defined in **NF**, the text is engraved before the character.

> **i** The engraving cycles are not available in **Machine** operating mode.

Character

Lower case

NF	Character
97	a
98	b
99	c
100	d
101	e
102	f
103	g
104	h
105	i
106	j
107	k
108	l
109	m
110	n
111	o
112	p
113	q
114	r
115	s
116	t
117	u
118	v
119	w
120	x
121	y
122	z

Upper case

NF	Character
65	A
66	B
67	C
68	D
69	E
70	F
71	G
72	H
73	I
74	J
75	K
76	L
77	M
78	N
79	O
80	P
81	Q
82	R
83	S
84	T
85	U
86	V
87	W
88	X
89	Y
90	Z

Numbers

NF	Character
48	0
49	1
50	2
51	3
52	4
53	5
54	6
55	7
56	8
57	9

Umlauts

NF	Character
196	Ä
214	Ö
220	Ü
223	ß
228	ä
246	ö
252	ü

Special characters

NF	Character	Meaning
32		Space
37	%	Per cent sign
40	(	Opening parenthesis
41	)	Closing parenthesis
43	+	Plus sign
44	,	Comma
45	-	Minus sign
46	.	Point
47	/	Forward slash
58	:	Colon
60	<	Less than character
61	=	Equal sign
62	>	Greater than character
64	@	At sign
91	[	Opening brackets
93	]	Closing brackets
95	_	Underscore
8364	€###	Euro sign
181	µ	Mu character
186	°	Degrees
215	*	Multiplication sign
33	!	Exclamation point
38	&	Ampersand and
63	?	Question mark
174	®	Trademark
216	Ø	Diameter sign

7.9 Drilling and milling patterns

$\mathbf{i}$ Note on using drilling and milling patterns:

- **Drilling patterns:** The control generates the machine commands **M12**, **M13** (apply and release shoe brake) under the following conditions: the drill/tap must be entered as driven tool (parameter **Tool driven, no=0/ yes=1 AW**) and the direction of rotation must be defined (parameter **Rotatnl. dir. M3=3, M4=4 MD**
- **ICP milling contours:** If the contour starting point is not on the origin of the coordinate system, the distance between the contour starting point and the coordinate origin is added to the pattern position
  Further information: "Examples of pattern machining", Page 430

Linear drilling pattern, axial

▶ Select **Drilling**

▶ Select **Drilling axial**

▶ Alternative: Select **Deep drilling axial**

▶ Alternative: Select **Tapping axial**
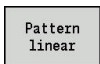
Pattern
linear

▶ Press the **Pattern linear** soft key

Press **Pattern linear** to machine hole patterns in which the individual features are arranged regularly spaced and in a straight line on the front face.

Cycle parameters:

- **X**, **Z**: **Start point**
- **C**: **Spindle angle** – C-axis position (default: current spindle angle)
- **Q**: **Number of holes**
- **X1**, **C1**: **Polar starting point** – Starting point of the pattern
- **XK**, **YK**: **Cartesian starting pnt**
- **I**, **J**: **End point (XK)** and **(YK)** – End point of the pattern (in Cartesian coordinates)
- **Ii**, **Ji**: **Distance (XKi)** and **(YKi)** – Incremental pattern spacing

The parameters of the hole are also requested.

Use the following parameter combinations to define the:
- Starting point of pattern:
  - **X1**, **C1**
  - **XK**, **YK**
- Pattern positions:
  - **Ii**, **Ji** and **Q**
  - **I**, **J** and **Q**

Cycle run:

1 Positioning (machine-dependent):
  - Without C axis: Position to the **Spindle angle C**
  - With C axis: Activate the C axis and position to the **Spindle angle C** at rapid traverse
  - In **Machine** operating mode: Machining from the current spindle angle

2 Calculate the pattern positions

3 Position the tool to the **Start point** of the pattern

4 Execute the drilling operation

5 Position for the next machining operation

6 Repeat 4 to 5 until all machining operations have been completed

7 Return to the **Start point**

8 Move to the **Tool change point** according to the setting in **G14**

Linear drilling pattern, radial

▶ Select **Drilling**

▶ Select **Drilling radial**

▶ Alternative: Select **Deep drilling radial**

▶ Alternative: Select **Tapping radial**

▶ Press the **Pattern linear** soft key

Press **Pattern linear** during drilling cycles to machine hole patterns in which the individual features are arranged regularly spaced in a straight line on the lateral surface.

Cycle parameters:

- **X**, **Z**: **Start point**
- **C**: **Spindle angle** – C-axis position (default: current spindle angle)
- **Q**: **Number of holes**
- **Z1**: **Start point pattern** – Position of first hole
- **ZE**: **End point pattern** (default: **Z1**)
- **C1**: **Angle of 1st hole** – Starting angle
- **Wi**: **Angle increment** – Pattern spacing (default: holes are regularly spaced on the lateral surface)

Define the pattern positions with **End point pattern** and **Angle increment** or **Angle increment** and **Number of holes**.

The parameters of the hole are also requested.

Cycle run:

1 Positioning (machine-dependent):
- Without C axis: Position to the **Spindle angle C**
- With C axis: Activate the C axis and position to the **Spindle angle C** at rapid traverse
- In **Machine** operating mode: Machining from the current spindle angle
2 Calculate the pattern positions
3 Position the tool to the **Start point** of the pattern
4 Execute the drilling operation
5 Position for the next machining operation
6 Repeat 4 to 5 until all machining operations have been completed
7 Return to the **Start point Z**
8 Move to the **Tool change point** according to the setting in **G14**

Linear milling pattern, axial

▶ Select **Milling**

▶ Select **Axial slot**

▶ Alternative: Select **Axial contour ICP**

Pattern linear

▶ Press the **Pattern linear** soft key

Press **Pattern linear** to machine milling patterns in which the individual features are arranged regularly spaced in a straight line on the front face.

Cycle parameters:

- **X**, **Z**: **Start point**
- **C**: **Spindle angle** – C-axis position (default: current spindle angle)
- **Q**: **Number of slots**
- **X1**, **C1**: **Polar starting point** – Starting point of the pattern
- **XK**, **YK**: **Cartesian starting pnt**
- **I**, **J**: **End point (XK)** and **(YK)** – End point of the pattern (in Cartesian coordinates)
- **Ii**, **Ji**: **Distance (XKi)** and **(YKi)** – Incremental pattern spacing

The parameters of the milling operation are also requested.

Use the following parameter combinations to define the:

- Starting point of pattern:
  - **X1**, **C1**
  - **XK**, **YK**
- Pattern positions:
  - **Ii**, **Ji** and **Q**
  - **I**, **J** and **Q**

Cycle run:

1 Positioning (machine-dependent):
- Without C axis: Position to the **Spindle angle C**
- With C axis: Activate the C axis and position to the **Spindle angle C** at rapid traverse
- In **Machine** operating mode: Machining from the current spindle angle

2 Calculate the pattern positions

3 Position the tool to the **Start point** of the pattern

4 Execute the milling operation

5 Position for the next machining operation

6 Repeat 4 to 5 until all machining operations have been completed

7 Return to the **Start point**

8 Move to the **Tool change point** according to the setting in **G14**

Linear milling pattern, radial

▶ Select **Milling**

▶ Select **Radial slot**

▶ Alternative: Select **Radial contour ICP**

| Pattern |
| linear |

▶ Press the **Pattern linear** soft key

Press **Pattern linear** during milling cycles to machine milling patterns in which the individual features are arranged regularly spaced in a straight line on the lateral surface.

Cycle parameters:

- **X**, **Z**: **Start point**
- **C**: **Spindle angle** – C-axis position (default: current spindle angle)
- **Q**: **Number of slots**
- **Z1**: **Start point pattern** – Position of first slot
- **ZE**: **End point pattern** (default: **Z1**)
- **C1**: **Start. angle** – Angle of first slot
- **Wi**: **Angle increment** – Pattern spacing (default: milling operations are regularly spaced on the lateral surface)

Define the pattern positions with **End point pattern** and Angle increment or Angle increment and Number of holes.

The parameters of the milling operation are also requested.

Cycle run:

1 Positioning (machine-dependent):
  - Without C axis: Position to the **Spindle angle C**
  - With C axis: Activate the C axis and position to the **Spindle angle C** at rapid traverse
  - In **Machine** operating mode: Machining from the current spindle angle
2 Calculate the pattern positions
3 Position the tool to the **Start point** of the pattern
4 Execute the milling operation
5 Position for the next machining operation
6 Repeat 4 to 5 until all machining operations have been completed
7 Return to the **Start point Z**
8 Move to the **Tool change point** according to the setting in **G14**

Circular drilling pattern, axial

▶ Select **Drilling**

▶ Select **Drilling axial**

▶ Alternative: Select **Deep drilling axial**
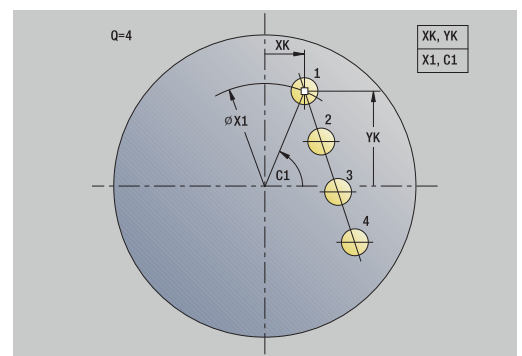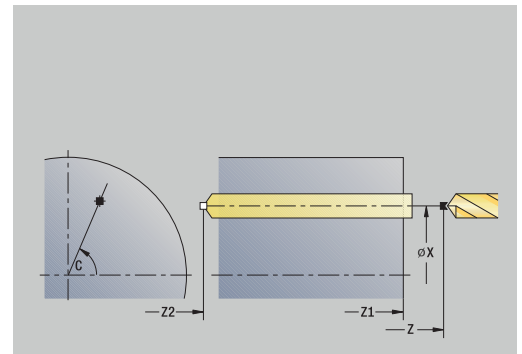
▶ Alternative: Select **Tapping axial**

Pattern
circular

▶ Press the **Pattern circular** soft key

Press **Pattern circular** during drilling cycles to machine drilling patterns in which the individual features are arranged regularly spaced in a circle or circular arc on the front face.

Cycle parameters:

- **X**, **Z**: **Start point**
- **C**: **Spindle angle** – C-axis position (default: current spindle angle)
- **Q**: **Number of holes**
- **XM**, **CM**: **Polar center point**
- **XK**, **YK**: **Cartesian center point**
- **K**: **Pattern diameter**
- **A**: **Angle of 1st hole** (default: 0°)
- **Wi**: **Angle increment** – Pattern spacing (default: holes are arranged regularly spaced in a circle)

The parameters of the hole are also requested.

Use the following parameter combinations to define the center of the pattern:

- **XM**, **CM**
- **XK**, **YK**

Cycle run:

1 Positioning (machine-dependent):

- Without C axis: Position to the **Spindle angle C**
- With C axis: Activate the C axis and position to the **Spindle angle C** at rapid traverse
- In **Machine** operating mode: Machining from the current spindle angle

2 Calculate the pattern positions

3 Position the tool to the **Start point** of the pattern

4 Execute the drilling operation

5 Position for the next machining operation

6 Repeat 4 to 5 until all machining operations have been completed

7 Return to the **Start point**

8 Move to the **Tool change point** according to the setting in **G14**

Circular drilling pattern, radial

▶ Select **Drilling**

▶ Select **Drilling radial**

▶ Alternative: Select **Deep drilling radial**

▶ Alternative: Select **Tapping radial**

▶ Press the **Pattern circular** soft key

Press **Pattern circular** during drilling cycles to machine hole patterns in which the individual features are arranged regularly spaced in a circle or circular arc on the lateral surface.
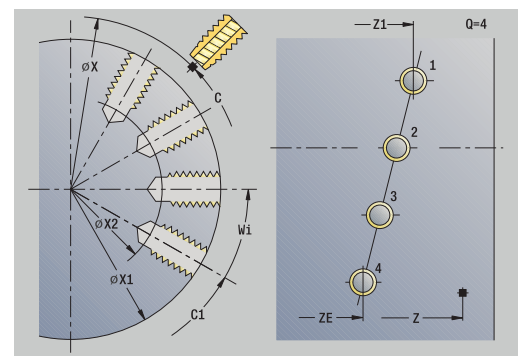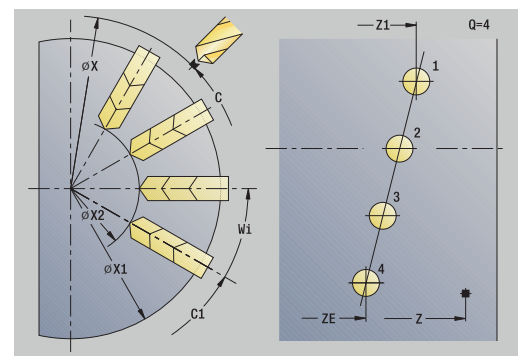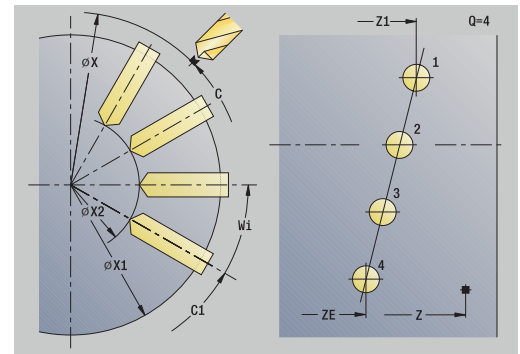
Cycle parameters:

- **X**, **Z**: **Start point**
- **C**: **Spindle angle** – C-axis position (default: current spindle angle)
- **Q**: **Number of holes**
- **ZM**, **CM**: **Center pt. in Z**, **Angle of pattern center**
- **K**: **Pattern diameter**
- **A**: **Angle of 1st hole** (default: 0°)
- **Wi**: **Angle increment** – Pattern spacing (default: holes are arranged regularly spaced in a circle)

The parameters of the hole are also requested.

Cycle run:

1 Positioning (machine-dependent):
- Without C axis: Position to the **Spindle angle C**
- With C axis: Activate the C axis and position to the **Spindle angle C** at rapid traverse
- In **Machine** operating mode: Machining from the current spindle angle
2 Calculate the pattern positions
3 Position the tool to the **Start point** of the pattern
4 Execute the drilling operation
5 Position for the next machining operation
6 Repeat 4 to 5 until all machining operations have been completed
7 Return to the **Start point**
8 Move to the **Tool change point** according to the setting in **G14**
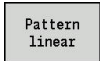
Circular milling pattern, axial

▶ Select **Milling**

▶ Select **Axial slot**

▶ Alternative: Select **Axial contour ICP**

Pattern circular

▶ Press the **Pattern circular** soft key

Press **Pattern circular** during milling cycles to machine milling patterns in which the individual features are arranged regularly spaced in a circle or circular arc on the front face.

Cycle parameters:

- **X**, **Z**: **Start point**
- **C**: **Spindle angle** – C-axis position (default: current spindle angle)
- **Q**: **Number of slots**
- **XM**, **CM**: **Polar center point**
- **XK**, **YK**: **Cartesian center point**
- **K**: **Pattern diameter**
- **A**: **Angle of 1st slot** (default: 0°)
- **Wi**: **Angle increment** – Pattern spacing (default: milling operations are arranged regularly spaced in a circle)

The parameters of the milling operation are also requested.

Use the following parameter combinations to define the:

- **XM**, **CM**
- **XK**, **YK**

Cycle run:

1 Positioning (machine-dependent):
   - Without C axis: Position to the **Spindle angle C**
   - With C axis: Activate the C axis and position to the **Spindle angle C** at rapid traverse
   - In **Machine** operating mode: Machining from the current spindle angle
2 Calculate the pattern positions
3 Position the tool to the **Start point** of the pattern
4 Execute the milling operation
5 Position for the next machining operation
6 Repeat 4 to 5 until all machining operations have been completed
7 Return to the **Start point**
8 Move to the **Tool change point** according to the setting in **G14**

Circular milling pattern, radial

▶ Select **Milling**

▶ Select **Radial slot**

▶ Alternative: Select **Radial contour ICP**

▶ Press the **Pattern circular** soft key

Press **Pattern circular** during milling cycles to machine milling patterns in which the individual features are arranged regularly spaced in a circle or circular arc on the lateral surface.

Cycle parameters:

- **X**, **Z**: **Start point**
- **C**: **Spindle angle** – C-axis position (default: current spindle angle)
- **Q**: **Number of slots**
- **ZM**, **CM**: **Center pt. in Z**, **Angle of pattern center**
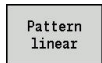- **K**: **Pattern diameter**
- **A**: **Angle of 1st slot** (default: 0°)
- **Wi**: **Angle increment** – Pattern spacing (default: milling operations are arranged regularly spaced in a circle)

The parameters of the milling operation are also requested.

ⓘ The starting point of an ICP contour to be arranged in a pattern must lie on the positive XK axis.

Cycle run:

1. Positioning (machine-dependent):
   - Without C axis: Position to the **Spindle angle C**
   - With C axis: Activate the C axis and position to the **Spindle angle C** at rapid traverse
   - In **Machine** operating mode: Machining from the current spindle angle
2. Calculate the pattern positions
3. Position the tool to the **Start point** of the pattern
4. Execute the milling operation
5. Position for the next machining operation
6. Repeat 4 to 5 until all machining operations have been completed
7. Return to the **Start point**
8. Move to the **Tool change point** according to the setting in **G14**
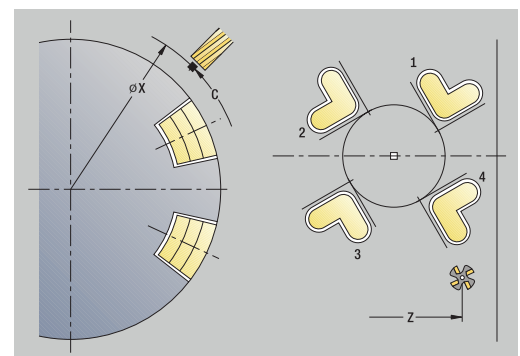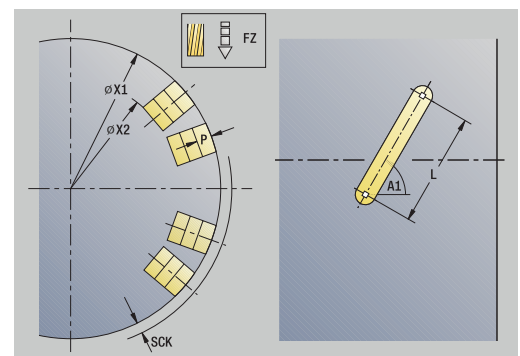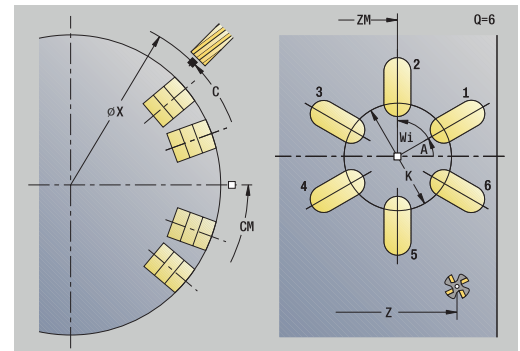
Examples of pattern machining

Linear hole pattern on face

A linear hole pattern is to be machined on the face of the workpiece with the **Drilling radial** cycle. This machining operation requires a traversable spindle and driven tools.

The pattern is programmed by entering the coordinates of the first and last hole, and the number of holes. Only the depth is indicated for the drilling cycle.

Tool data

- **TO** = 8 – Tool orientation
- **DV** = 5 – Drilling diameter
- **BW** = 118 – Point angle
- **AW** = 1 – The tool is a driven tool
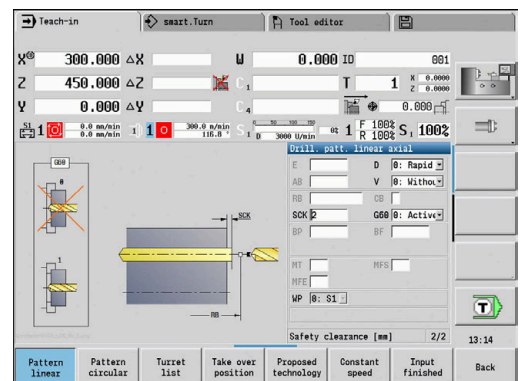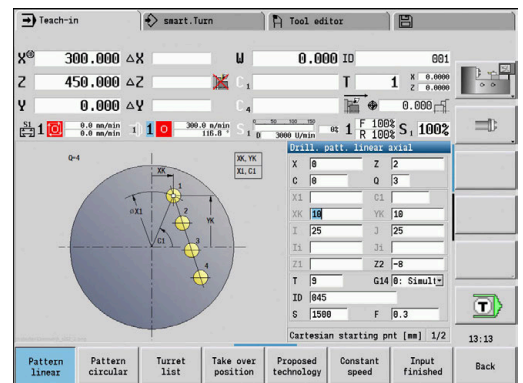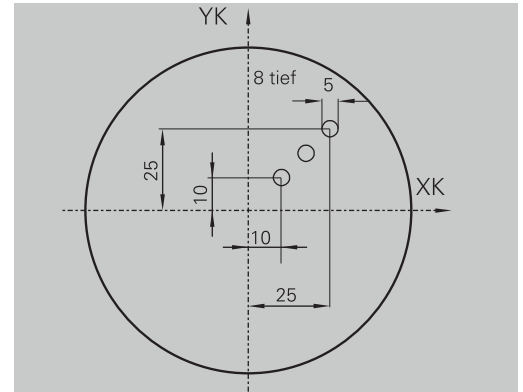
Circular hole pattern on face

A circular hole pattern is to be machined on the face of the workpiece with the **Drilling axial** cycle. This machining operation requires a traversable spindle and driven tools.

The **Center point pattern** is specified in Cartesian coordinates.

Since this example is to illustrate how you drill a through hole, the **End point drill Z2** is programmed such that the tool has to drill all the way through the workpiece before it reaches the end point. The **AB** and **V** parameters define a feed-rate reduction for both spot drilling and drilling through-holes.

Tool data

- **TO** = 8 – Tool orientation
- **DV** = 5 – Drilling diameter
- **BW** = 118 – Point angle
- **AW** = 1 – The tool is a driven tool
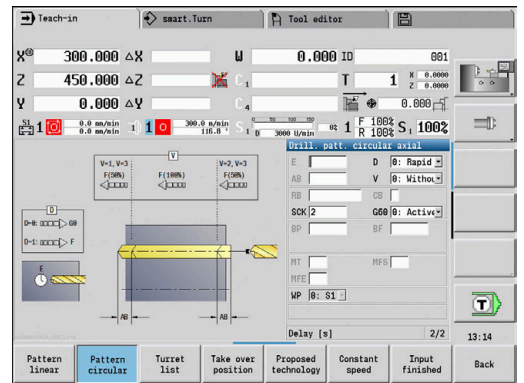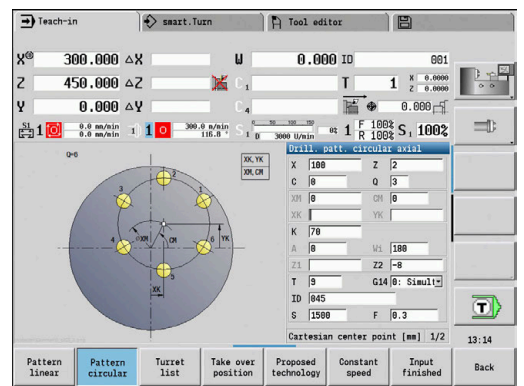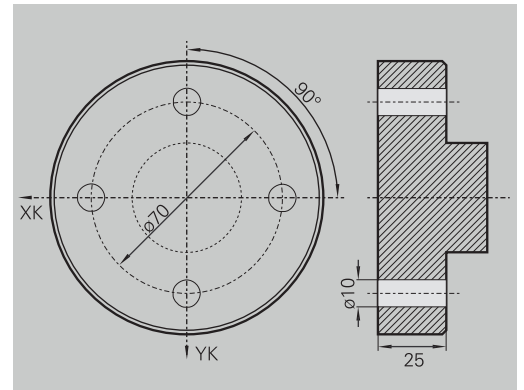
Linear hole pattern on lateral surface

A linear hole pattern is to be machined on the lateral surface of the workpiece with the **Drilling axial** cycle. This machining operation requires a traversable spindle and driven tools.

The drilling pattern is defined by the coordinates of the first hole, the number of holes, and the spacing between the holes. Only the depth is indicated for the drilling cycle.

Tool data

- **TO** = 2 – Tool orientation
- **DV** = 8 – Drilling diameter
- **BW** = 118 – Point angle
- **AW** = 1 – The tool is a driven tool
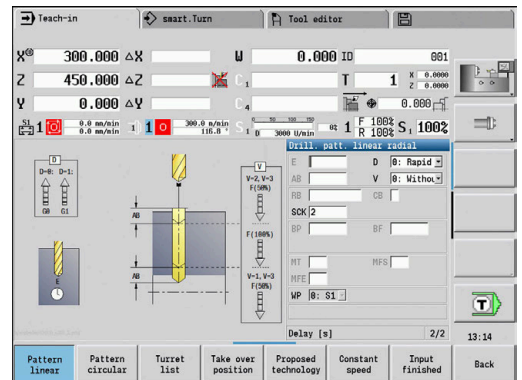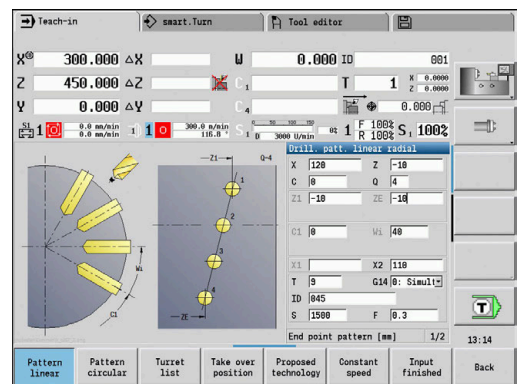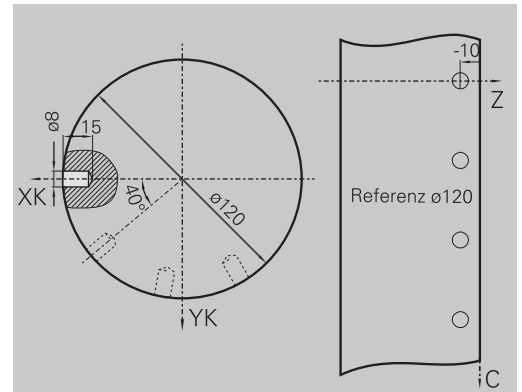
7.10 DIN cycles

Menu item	Meaning
DIN	This function allows you to select a DIN cycle (DIN subprogram) and integrate it in a cycle program. The dialogs of the parameters defined in the subprogram are then shown in the form.

The technology data that are programmed in the DIN cycle (in **Machine** operating mode, the currently active technology data) becomes effective as soon as you start the DIN subprogram. However, you can change the machine data (**T, S, F**) at any time by editing the DIN subprogram.

DIN cycle

DIN	▶ Select **DIN cycle**

Cycle parameters:

- **L**: **DIN subprogram** – DIN macro number
- **Q**: **Number of repetitions** (default: 1)
- **LA-LF**: **Transmiss.value**
- **LH-LK**: **Transmiss.value**
- **LO-LP**: **Transmiss.value**
- **LR-LS**: **Transmiss.value**
- **LU**: **Transmiss.value**
- **LW-LZ**: **Transmiss.value**
- **LN**: **Transmiss.value**
- **T**: **Tool number** – turret pocket number
- **ID**: **ID no.**
- **S**: **Cutting speed** or **Constant speed**
- **F**: **Feed per revolution**
- **MT**: **M after T**: **M** function that is executed after the **T** tool call
- **MFS**: **M at beginning**: **M** function that is executed at the beginning of the machining step
- **MFE**: **M at end**: **M** function that is executed at the end of the machining step
- **WP**: **No. of spindle** – Displays which workpiece spindle is used to execute the cycle (machine-dependent)
  - Main drive
  - Opposing spindle for rear-face machining
- **BW**: **Angle in the B axis** (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
- **HC**: **Shoe brake** (machine-dependent)
- **DF**: **Miscellaneous function** (machine-dependent)
- **ID1**, **AT1**: **ID number**
- **BS**, **BE**, **WS**, **AC**, **WC**, **RC**, **IC**, **KC**, **JC**: **Transmiss.value**

Type of machining for technology database access depends on the tool type:

1 Turning tool: **Roughing**

2 Button tool: **Roughing**

3 Threading tool: **Thread cutting**

4 Recessing tool: **Contour recessg.**

5 Twist drill: **Drilling**

6 Indexable insert drill: **Predrilling**

7 Tap: **Tapping**

8 Milling cutter: **Milling**

> ℹ️ In the DIN subprogram, you can assign texts and help graphics to the transfer values.
>
> **Further information**: smart.Turn and DIN Programming User's Manual

---

NOTICE

Danger of collision!

DIN cycles do not contain any starting points. When a DIN cycle is called, the control positions the tool on a diagonal path from the current position to the first position that is programmed in the DIN subprogram. There is danger of collision during the approach movement!

▶ Before calling the DIN cycle, pre-position the tool if necessary

---

NOTICE

Danger of collision!

In **Teach-in** submode, all datum shifts contained in a DIN cycle (DIN macro) will be reset after the cycle has been executed. Danger of collision during subsequent machining operations!

▶ Use DIN cycles without datum shifts

8

ICP Programming

8.1 ICP contours

The interactive contour programming (**ICP**) feature provides graphic support when you are defining the workpiece contours. (**ICP** is the abbreviation of **I**nteractive **C**ontour **P**rogramming.)

Contours created with **ICP** are used in the following:

- In the ICP cycles (**Teach-in** submode, **Machine** operating mode)
- In **smart.Turn** mode of operation

Each contour begins with a starting point. The following contours are defined using linear and circular contour elements as well as form elements like chamfers, rounding radii, and undercuts.

ICP is called from the **smart.Turn** mode of operation and the cycle dialogs.

ICP contours created in Teach-in mode are saved by the control in independent files. You can enter file names (contour names) with up to 40 characters. ICP contours are integrated in ICP cycles.

The following contour types are possible:

- Turning contours: ***.gmi**
- Contours of workpiece blanks: ***.gmr**
- Milling contours on face: ***.gms**
- Milling contours on lateral surface: ***.gmm**

The control integrates **ICP contours** created in **smart.Turn** operating mode into the respective NC program. Contour descriptions are saved as **G** codes.
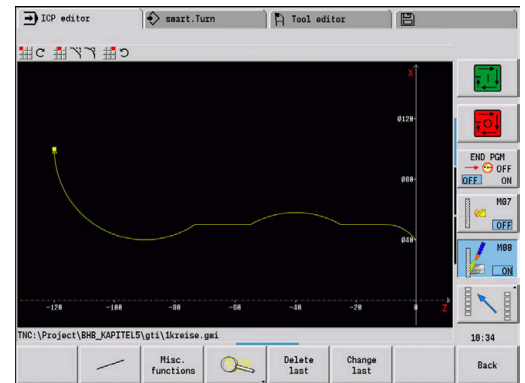
> ⓘ
> - In Teach-in mode, **ICP contours** created are managed in independent files. These contours can only be edited with **ICP**
> - In **smart.Turn** mode of operation, contours are part of the NC program. They can be machined with the ICP editor or **smart.Turn** editor.

> ⓘ
> In the **convertICP** (602023) machine parameter, you define whether the control loads the programmed or calculated values into the NC program.

Loading contours

ICP contours that you have created for cycle programs can be loaded in **smart.Turn** operating mode. **ICP** converts the contours into **G** codes and integrates them in the smart.Turn program. The contour is now part of the smart.Turn program.

Contours that exist in DXF format can be imported with the **ICP editor** submode. During the import process, the contours are converted from DXF format to ICP format. You can use DXF contours both for the **Teach-in** submode and for the **smart.Turn** operating mode.

Form elements

You can insert chamfers and rounding radii at each corner of the contour.

Undercuts according to DIN 76, DIN 509 E, and DIN 509 F can be inserted at paraxial, orthogonal contour corners, and the control tolerates small deviations in horizontal elements (X direction).

You have the following alternatives for entering form elements:

- Enter all contour elements, including the form elements, in the sequence in which they are given in the workpiece drawing.
- First define the rough contour without the form elements. Then you **superimpose** the form elements.
  Further information: "Superimposing form elements", Page 458

Machining attributes

You can assign the following machining attributes to the contour elements.

Machining attributes:

- **U**: **Oversz** is additive to other oversizes
  ICP generates a **G52 Pxx H1**.
- **F**: **Feed per rot.** – (special feed rate for finishing)
  ICP generates a **G95 Fxx**.
- **D**: **Corr. additiv** – Number of the additive D compensation for the finishing operation, e.g. **D** = 01 to 16
  ICP generates a **G149 D9xx**.
- **FP**: **Do not machine the element** (required only for **TURN PLUS**)
  - **0: No**
  - **1: Yes**
- **IC**: **Measuring cut oversize** (not available in **Teach-in** submode)
- **KC**: **Length of measuring cut** (not available in **Teach-in** submode)
- **HC**: **Measuring cut counter** – Number of workpieces after which a measurement is performed

> **i** The machining attributes are effective only for the element in which the attributes have been entered in **ICP**.

Calculation of contour geometry

The control automatically calculates all missing coordinates, points of intersection, center points, etc. that can be derived mathematically.

If the entered data permit several mathematically possible solutions, you can inspect the individual solutions and select the proposal that matches the drawing.

Each unresolved contour element is represented by a small symbol below the graphic window. The control displays all contour elements that can be drawn, even if they are not yet fully defined.
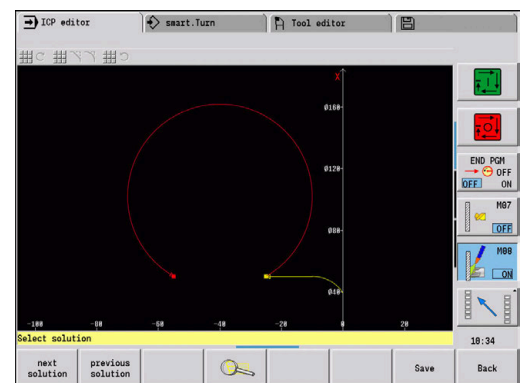
8.2 ICP editor submode in Teach-in

In Teach-in mode you can create:

- Complex workpiece blank contours
- Contours for turning
    - For ICP turning cycles
    - For ICP recessing cycles
    - For ICP recess-turning cycles
- Complex contours for milling with the C axis
    - For the face
    - For the lateral surface

Activate the **ICP editor** submode with the **Edit ICP** soft key. It can only be selected when editing ICP cutting cycles or ICP milling cycles or with the ICP workpiece blank contour cycle.

The description depends on the contour type. Using the cycle, **ICP** distinguishes between:

- Contour for turning or the workpiece blank contour:
  Further information: "Contour elements – turning contour", Page 468
- Contour for the face:
  Further information: "Front face contours in smart.Turn operating mode (option 9)", Page 493
- Contour for the lateral surface:
  Further information: "Lateral surface contours in smart.Turn operating mode (option 9)", Page 502

> **i** If you create or edit two or more **ICP contours** in sequence, the last edited **ICP contour number** is loaded to the cycle after the **ICP editor** submode has been exited.
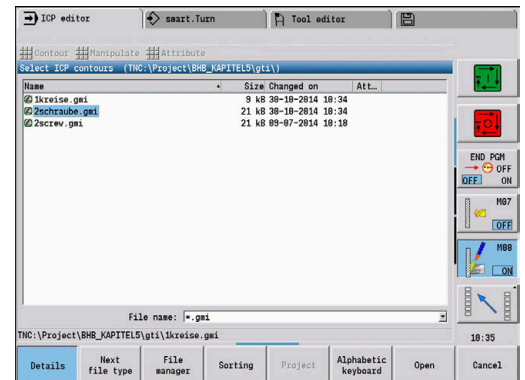
Editing contours for cycles

The **ICP contours** of the cycle editing function are assigned names. The contour name is the same as the file name. The contour name is also used in the calling cycle.

You can define the contour name in the following ways:

- Define the contour name in the cycle dialog **before** calling the **ICP editor** submode (**FK** input field). **ICP** loads this name
- Define the contour name in the **ICP editor** submode. The input field **FK** must be empty when you call the **ICP editor** submode
- Load the existing contour. When you exit the **ICP editor** submode, the name of the contour last edited is loaded into the **FK** input field

To create a new contour:

Edit ICP	▶ Define the contour name in the cycle dialog and press the **Edit ICP** soft key. The **ICP editor** submode switches to contour input
Edit ICP	▶ Alternative: Press the **Edit ICP** soft key. The **ICP editor** submode opens the **Select ICP contours** window
Open	▶ Define the contour name in the **File name:** field and press the **Open** soft key. The **ICP editor** submode switches to entering the contour
	▶ Select the **CONTOUR** menu item
Insert element	▶ Press the **Insert element** soft key
	> **ICP** waits for you to enter a new contour

File management in ICP editor submodeICP editor

In file management, you can copy, rename, or delete the **ICP contours**.

To open the file manager:

Edit ICP	▶ Press the **Edit ICP** soft key
Contour list	▶ Press the **Contour list** soft key
	> The **ICP editor** submode opens the **Select ICP contours** window
File manager	▶ Press the **File manager** soft key
	> The **ICP editor** submode switches the soft-key row to the file management functions

8.3 ICP editor submode in smart.Turn operating mode

In **smart.Turn** mode of operation you create:

- Contour groups
- Workpiece blank contours and auxiliary workpiece blank contours
- Finished part contours and auxiliary contours
- Standard figures and complex contours for C-axis machining
  - on the face
  - on the lateral surface
- Standard figures and complex contours for Y-axis machining
  - on the XY plane
  - on the YZ plane

Contour groups: The control supports up to four contour groups (**BLANK**, **FINISHED PART**, and **TEMPORARY**) in one NC program. The **Contour group** section code initiates the description of a contour group.

Further information: "Contour groups", Page 541

Workpiece blank contours and auxiliary workpiece blank contours: You can describe complex workpiece blanks element by element —just like finished parts. Select the standard forms of bar and tube from the menu and describe them with a few parameters. If a definition of the finished part exists, you can also select Cast part in the menu.

Further information: "Workpiece blank definition", Page 467

Figures and patterns for C and Y axis machining: You describe complex milling contours element by element. The following standard figures are prepared.

You select the figures by menu and describe them with a few parameters:

- Circle
- Rectangle
- Polygon C
- Linear slot
- Circular slot
- Hole

You can arrange these figures and holes as linear or circular patterns on the face or lateral surface as well as in the XY or YZ plane.

You can import **DXF contours** and integrate them in the smart.Turn program.

Cycle programming contours can be loaded and integrated into the smart.Turn program.

The **smart.Turn** operating mode supports loading the following contours:

- Workpiece blank description (extension ***.gmr**): Loaded as workpiece blank contour or auxiliary workpiece blank contour
- Contour for turning (extension ***.gmi**): Loaded as finished part contour or auxiliary contour
- Face contour (extension ***.gms**)
- Lateral surface contour (extension ***.gmm**)

> **i** **ICP** uses **G** codes to describe the contours created in the smart.Turn program.
>
> In the **convertICP** (602023) machine parameter, you define whether the control loads the programmed or calculated values into the NC program.

Editing contours for cycles

Programming a new workpiece blank contour:

▶ Select the **ICP** menu item

▶ Select **Workpiece blank** or **New auxiliary blank** in the ICP submenu

▶ Select the **Contour** menu item

> The **ICP editor** submode switches to the editor that allows you to enter the complex workpiece blank contour

▶ Alternative: Select the **Bar** menu item

> Describe the standard workpiece blank **Bar**

▶ Alternative: Select the **Tube** menu item

> Describe the **Tube** standard workpiece blank
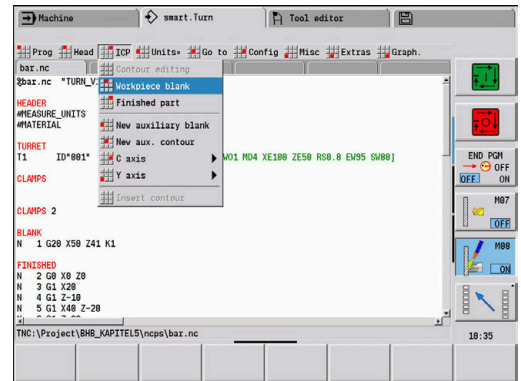
Programming a new contour for turning:

▶ Select the **ICP** menu item

▶ Select the contour type in the ICP submenu

▶ Select the **Contour** menu item

▶ Alternative: Press the **Insert element** soft key

> **ICP** waits for you to enter a new contour

Loading a contour from cycle editing:

▶ Select the **ICP** menu item

▶ Select the contour type in the ICP submenu

▶ Press the **Contour list** soft key

> In **ICP editor** submode, you can display the list of the contours created in Teach-in mode

▶ Select and load the contour

Editing an existing contour:

▶ Select the **ICP** menu item

▶ Select **Contour editing** in the ICP submenu

▶ Alternative: Press the **Change ICP contour** soft key

> The **ICP editor** submode displays the selected existing contour for subsequent editing

8.4   Programming ICP contours

An ICP contour consists of individual contour elements. Create the contour by entering the individual contour elements one after the other in the correct sequence. The **Start point** is defined before you describe the first contour element. The **Final point** is determined by the target point of the last contour element.

The contour elements and subcontours are displayed as soon as they are programmed. With the zoom and panning functions, you can adjust the graphics as required.

The principle described in the following applies to all **ICP contours**.

Soft keys in the ICP editor submode – Main menu

Contour list	Open the file selection dialog for **ICP contours**
Turn contour	Invert the definition direction of the contour
	Retroactively insert form elements
	Open the soft-key menu for the zoom functions and display the zoom frame
Delete element	Clear an existing element
Change element	Modify an existing element
Insert element	Add an element to the existing contour
Back	Return to the dialog that has called **ICP**

Defining an ICP contour

When you create a new contour, the control first asks for the coordinates of the **contour starting point**.

Linear contour elements: Use the menu symbol to select the direction of the element and assign it a dimension. When defining horizontal and vertical linear elements, it is not necessary to enter the X and Z coordinates, provided that there are no unresolved elements.

Circular contour elements: Use the menu symbol to select the direction of arc rotation and dimension the arc.

After selecting a contour element, enter the known parameters. The control automatically calculates parameters that have not been defined from the adjoining contour elements. Usually the contour elements are described using the dimensions given in the production drawing.

During input of linear or circular elements, the control does display the **Start point**, but it is not editable. The **Start point** is the **Final point** of the preceding element.

You can toggle between the **line and arc menus** by soft key. Form elements (chamfers, roundings, and undercuts) are selected using the menu item.

To program an ICP contour:

▶ Select the **Contour** menu item
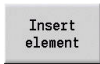
▶ Define the starting point
▶ Alternative: Press the **Insert element** soft key

▶ Define the starting point
▶ Press the **Line menu** soft key

▶ Alternative: Press the **arc menu** soft key

▶ Select the element type
▶ Enter the known parameters of the contour element

Menu items of the line menu
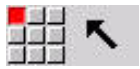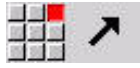
	Line with angle in displayed quadrant
	Horizontal line in displayed direction
	Line with angle in displayed quadrant
	Vertical line in displayed direction
	Call the form element menu
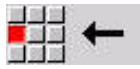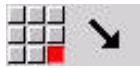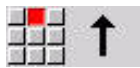
Menu items of arc menu

	Circular arc in the displayed direction of rotation
	Call the form element menu

Soft keys for switching between line menu and arc menu

	Press the **Line menu** soft key
	Press the **Arc menu** soft key

Absolute or incremental dimensions

The setting of the **Increment** soft key determines which type of coordinate is active for dimensioning. Incremental parameters will have the appendix **i** (**Xi**, **Zi**, etc.).

Increment soft key switchover

Increment	Activate the incremental dimension for the current value

Transitions between contour elements

A transition between two contour elements is called **tangential** when one contour element makes a smooth and continuous transition to the next. There is no visible kink or corner at the intersection. With geometrically complex contours, tangential transitions are useful for reducing the input of dimensional data to a minimum and eliminating the possibility of mathematically contradictory entries.

To be able to calculate unresolved contour elements, the control must know the type of transition that connects the contour elements. The transition to the next contour element is determined by soft key.

> **ℹ** Error messages that occur during definition of the ICP contour are often caused by **forgotten** tangential transitions.

Soft key for tangential transition

	Activates the tangential condition for the transition in the end point of the contour element

Fits and internal threads

With the **Fit Inside thd.** soft key, you can display an input form for calculating the machining diameter for fits and internal threads. After entering the required values (nominal diameter and tolerance class or thread type), you can load the calculated value as the target point for the contour element.

> **i** The machining diameter can only be calculated for suitable contour elements, e.g. for a linear element in X direction when calculating the fit for a shaft.
>
> When calculating internal threads of thread type 9, 10 or 11, you can select the nominal diameter for inch threads in the **Nominal dia. from list L** list.

To calculate the fit for a hole or shaft:

Fit Inside thd.	► Press the **Fit Inside thd.** soft key
	► Enter the nominal diameter
	► Enter the fit data in the **Fit** form
ENT	► Press the **ENT** key to calculate the values
Take over	► Press the **Take over** soft key > The calculated center of tolerance is entered in the corresponding field of the open dialog.

To calculate the core hole diameter for internal threads:

Inner thread	► Press the **Inner thread** soft key
	► Enter the nominal diameter
	► Enter the thread data in the **Inside thread calculator** form
ENT	► Press the **ENT** key to calculate the values
Take over	► Press the **Take over** soft key > The calculated core hole diameter is entered in the corresponding field of the open dialog.

Polar coordinates

Entry of Cartesian coordinates is expected as standard. With the soft keys for polar coordinates you switch individual coordinates to polar coordinates.

You can mix Cartesian coordinates and polar coordinates to define a point.

Soft keys for polar coordinates

	Switch the field to entering the angle **W**
	Switch the field to entering the radius **P**

Angle input

Select the desired angle input by soft key.

- **Linear elements**
  - **AN** Angle to Z axis (**AN**<= 90° – Within the preselected quadrant)
  - **ANn** Angle to the following element
  - **ANp** Angle to the previous element
- **Circular arcs**
  - **ANs** Tangential angle in the starting point of the circle
  - **ANe** Tangential angle in the end point of the circle
  - **ANn** Angle to the following element
  - **ANp** Angle to the previous element
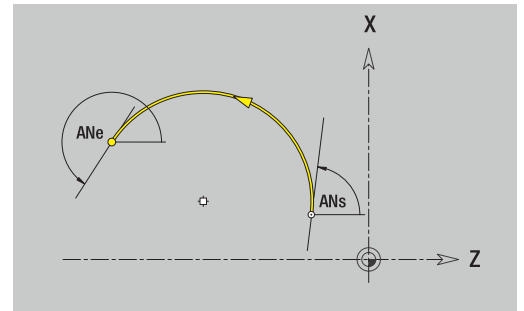
Soft keys for angular input

	Angle to successor
	Angle to predecessor

Contour graphics

As soon as you have entered a contour element, the control checks whether the element is "resolved" or "unresolved."

- A **resolved element** is a contour element that is fully and unambiguously defined. It is drawn immediately.
- An **unresolved element** has not yet been fully defined by the specified data. Using the **ICP editor**
  - It places a symbol below the graphic window that reflects the element type and the line direction / direction of rotation
  - It displays an unresolved linear element if the starting point and direction are known
  - It displays an unresolved circular element as a full circle if the circle center and the radius are known

As soon as it can be calculated, the control converts an unresolved contour element to a resolved contour element. The symbol is then deleted.

An incorrect contour element is displayed if possible. In addition, an error message is issued.

Unresolved contour elements: If during further contour input an error results because there is not enough information for a contour element, the unresolved element can be selected and the missing information added.

If a contour contains **unresolved** contour elements, the resolved elements cannot be changed. You can, however, set or delete the **tangential transition** for the contour element located directly before the unresolved contour area.

> ⓘ
> - If the element to be edited is an unresolved element, the associated symbol is marked **selected**
> - The element type and the direction of rotation of a circular arc cannot be changed. In this case, the contour element has to be deleted and then added

Selection of solutions

If the data entered for unresolved contour elements permit several possible solutions, you can use the **next solution** and **previous solution** soft keys to check all mathematically possible solutions. Confirm the correct solution by soft key.

> **i** If the contour still contains unresolved contour elements when you exit the editing mode, the control will ask you whether to discard these elements.
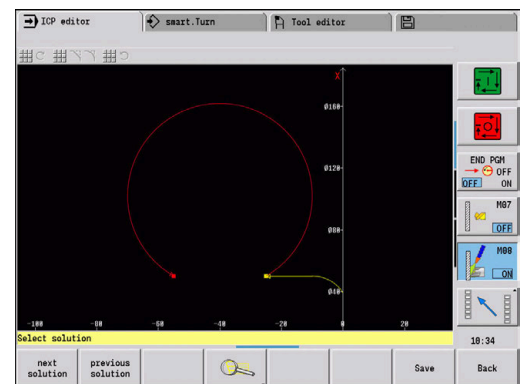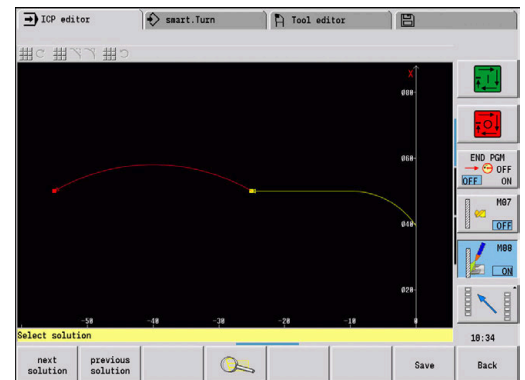
Colors in contour graphics

Resolved, unresolved, and selected contour elements as well as selected contour corners and remaining contours are depicted in different colors. (The selection of contour elements, contour corners, and remaining contours is important when editing **ICP contours**.)

Colors:

- White: Workpiece blank contour, auxiliary workpiece blank contour
- Yellow: Finished part contours (turned contours, contour for C and Y axis machining)
- Blue: Auxiliary contours
- Gray: For unresolved or incorrect but displayable elements
- Red: Selected solution, selected element or selected corner

Selection functions

In the **ICP editor** submode, the control provides various functions for selecting contour elements, form elements, contour corners, and contour areas. These functions can be called via soft key, touch gestures, or the mouse.

Selected contour corners or contour elements are shown in red.

To select a contour area:
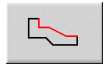
▶ Select the first element of the contour area

▶ Activate the area selection

▶ Press the **Next element** soft key until the entire area is marked

▶ Alternative: Press the **Previous element** soft key until the entire area is marked

---

Selecting contour elements

	Next element (or the left arrow key) selects the next element in the direction of contour definition
	Previous element (or the right arrow key) selects the previous element in the direction of contour definition
	Select area activates the area selection

Selecting contour corners (for form elements)

	Next contour corner (or the left arrow key) selects the next corner in the direction of contour definition
	Previous contour corner (or the right arrow key) selects the previous corner in the direction of contour definition
	Mark all corners marks all contour corners
	Corner selection If the corner selecting function is active, multiple contour corners can be marked
Mark	**Mark** If the corner selecting function is active, individual contour corners can be selected and marked or unmarked

Datum shift

With this function, you can move a complete turning contour.

First, select the Finished part menu:

▶ Select the **ICP** menu item

▶ Select the **Finished part** menu item

To activate a datum shift:

▶ Select the **Contour** menu item
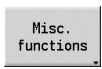
▶ Press the **Misc. functions** soft key

▶ Select the **Datum** menu item

▶ Select the **Shift** menu item

▶ Enter the contour shift to move the defined contour

▶ Press the **Save** soft key

To deactivate a datum shift:

▶ Select the **Datum** menu item

▶ Select the **Resetting** menu item
> The control resets the origin of the coordinate system to the originally programmed position.
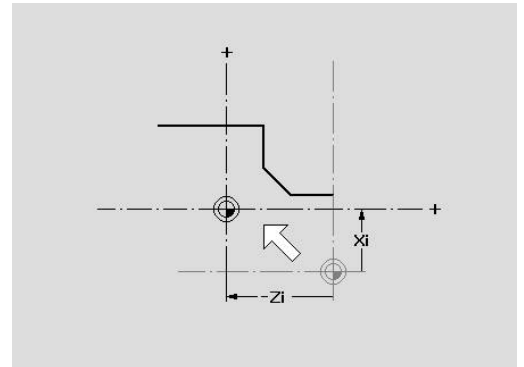
ℹ The datum shift cannot be reset after you have exited the **ICP editor** submode. When you exit the **ICP editor** submode, the contour is recalculated with the values relative to the datum shift and saved. The only thing you can do in this case is a datum shift in the opposite direction.

Parameters

- **Xi**: **Target point** – Value by which the datum is shifted
- **Zi**: **Target point** – Value by which the datum is shifted

Duplicating a contour section in linear series

With this function, you can define a contour section and append it to the existing contour.

First, select the Finished part menu:

▶ Select the **ICP** menu item

▶ Select the **Finished part** menu item

To duplicate:

▶ Select the **Contour** menu item
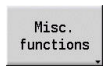
▶ Press the **Misc. functions** soft key

▶ Select the **Duplicate** menu item

▶ Select the **Linear row** menu item

▶ Use the **Next element** or **Previous element** soft key to select contour elements

▶ Press the **Select** soft key

▶ Enter the number of repeats

▶ Press the **Save** soft key

Parameters

■ **Q**: **Quantity** (number of repeats)
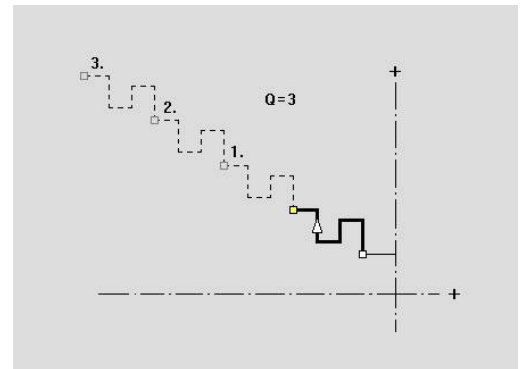
Duplicating a contour section in circular series

With this function, you can define a contour section and append it to the existing contour in a circular series.

First, select the Finished part menu:

▶ Select the **ICP** menu item

▶ Select the **Finished part** menu item

To duplicate:

▶ Select the **Contour** menu item
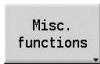
▶ Press the **Misc. functions** soft key

▶ Select the **Duplicate** menu item in the finished part menu

▶ Select the **Circular row** menu item

▶ Use the **Next element** or **Previous element** soft key to select contour elements

▶ Press the **Select** soft key

▶ Enter the number of repeats and the radius

▶ Press the **Save** soft key
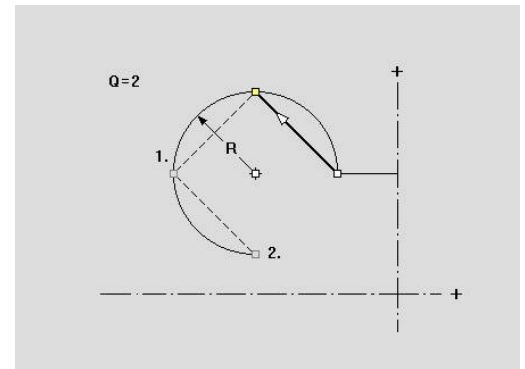
Parameters
- **Q**: **Quantity** – The contour section is duplicated **Q** times
- **R**: **Radius**

ℹ️ The control uses the defined radius for creating circles both around the starting point and the end point of the contour section. The points of intersection of the circles define the two possible centers of rotation.

The angle of rotation is calculated from the distance between the starting point and the end point of the contour section.

Use the **next solution** or **previous solution** soft key to select one of the mathematically possible solutions.

Duplicating a contour section by mirroring

With this function, you can define a contour section, mirror it and append it to the existing contour.

First, select the Finished part menu:

▶ Select the **ICP** menu item

▶ Select the **Finished part** menu item

To duplicate:

▶ Select the **Contour** menu item

▶ Press the **Misc. functions** soft key

▶ Select the **Duplicate** menu item

▶ Select the **Mirroring** menu item

▶ Use the **Next element** or **Previous element** soft key to select contour elements
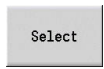
▶ Press the **Select** soft key

▶ Enter the angle of the mirror axis
▶ Press the **Save** soft key

Parameters
- **W**: **Angle of the mirror axis** – The mirror axis intersects the current end point of the contour (angle reference: positive Z axis)

Inverting

With the **Invert** function, you can reverse the programmed direction of a contour.
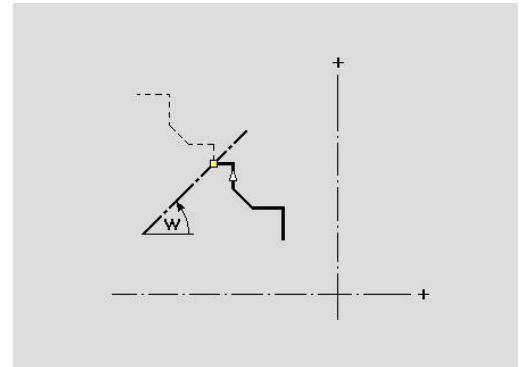
Contour direction (cycle programming)

The cutting direction during cycle programming depends on the direction of the contour. If the contour is described in the −Z direction, a tool with the orientation 1 must be used for longitudinal machining. The cycle used determines whether machining is transverse or longitudinal.

Further information: "General tool parameters", Page 588

If the contour is described in the −X direction, a transverse machining cycle or a tool with the orientation 3 must be used.

- **ICP cut, longitudinal/transverse (roughing)**: The control machines the workpiece in the contour direction.
- **ICP finishing, longitudinal/transverse**: The control finishes in the contour direction.

> **ℹ** An ICP contour which was defined for a roughing operation with the ICP cycle for longitudinal cutting cannot be used for machining with the ICP cycle for transverse cutting. For this purpose, reverse the contour direction with the **Turn contour** soft key.

Soft keys in ICP editor submode – Main menu

Turn contour	Invert the definition direction of the contour
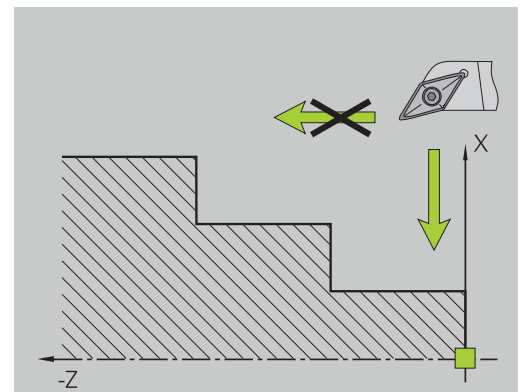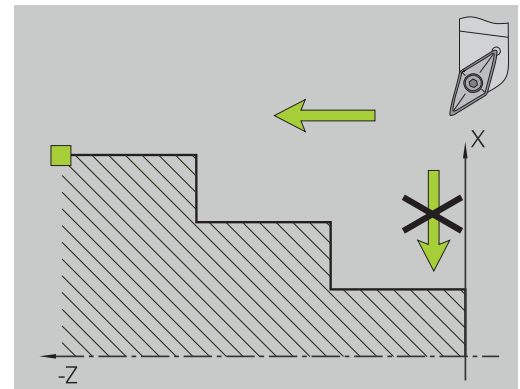
8.5   Editing ICP contours

The control offers the following possibilities for extending or changing a programmed contour.

Superimposing form elements

To superimpose form elements:

▶ Press the **Form elements** soft key

▶ Select the desired form element

▶ Select the corner

▶ Confirm the corner for the form element

▶ Enter the data for the form element

▶ Press the **Save** soft key
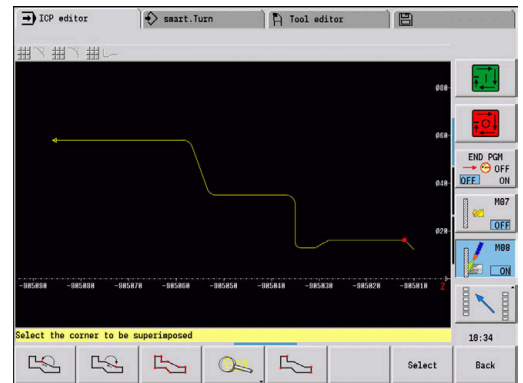
Adding contour elements

You can **add to** an ICP contour by entering additional contour elements that are **appended** to the existing contour. A small square indicates the end of the contour and an arrow indicates the direction.

To add contour elements:

▶ Press the **Insert element** soft key

▶ Append additional contour elements to the existing contour

Editing or deleting the last contour element

Editing the last contour element: When you press the **Change last** soft key, the data of the **last** contour element is made available for editing.

Depending on the adjoining contour elements, corrections of linear or circular elements are either transferred immediately or the corrected contour is displayed for inspection. **ICP** highlights the affected contour elements in color. If there are several possible solutions, use the **next solution** and **previous solution** soft keys to check all mathematically possible solutions.

The change will not become effective until you confirm by soft key. If you discard the change, the **old** description becomes effective again.

The type of contour element (linear or circular element), the direction of a linear element, or the direction of rotation of a circular element cannot be changed. Should this be necessary, you must delete the element and add a new contour element.

Deleting the last contour element: When you press the **Delete last** soft key, the data of the **last** contour element is discarded. You can use this function repeatedly to delete multiple successive contour elements.

Deleting contour elements

To delete a contour element:

▶ Select the **Manipulate** menu item

> The menu displays functions for trimming, editing, and deleting contours.

▶ Select the **Delete** menu item

▶ Select the **Element/range** menu item

▶ Select the contour element to be deleted

▶ Press the **Delete Yes** soft key

You can delete several successive contour elements.

Manipulating contour elements

The control provides various ways to change an existing contour. The procedure is illustrated in the following example of **editing the length of an element**. The other functions work similar to the procedure described here.

The **Manipulate** menu provides the following functions for editing existing contour elements:

- **Trimming**
    - **Length element**
    - **Contour length** (only for closed contours)
    - **Radius**
    - **Diameter**
- **Modify**
    - **Contour element**
    - **Form element**
- **Delete**
    - **Element/range**
    - **Move element/range as well**
    - **Contour/pocket/Figure/pattern**
    - **Form element**
    - **All form elements**
- **Transform**
    - **Shift** a contour
    - **Turning** a contour
    - **Mirroring** a contour: You can define the position of the mirror axis by entering the starting point and end point coordinates or the starting point coordinates and the angle

Changing the length of the contour element

To change the length of the contour element:

- ▶ Select the **Manipulate** menu item
- > The menu displays functions for trimming, editing, and deleting contours.

- ▶ Select the **Modify** menu item

- ▶ Select the **Contour element** menu item

- ▶ Select the contour element to be edited

- ▶ Present the selected contour element for editing

  Modify
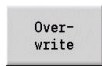
- ▶ Make the changes
- ▶ Load the changes

  Over-
  write

- > The contour or the possible solutions are displayed for inspection. Changes in form elements and unresolved elements are loaded immediately (original contour in yellow, changed contour in red for comparison).
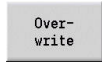
- ▶ Load the desired solution

  Over-
  write

Changing the paraxial line

When **changing** a paraxial line, an additional soft key is offered with which you can change the second end point as well. From an originally straight line you can make a diagonal in order to make corrections.

To change the paraxial line:

- ▶ Change the **fixed** end point. By selecting the menu item repeatedly, you can select the direction of the diagonal

Shifting a contour

To shift a contour:

▶ Select the **Manipulate** menu item
> The menu displays functions for trimming, editing, and deleting contours.

▶ Select the **Modify** menu item

▶ Select the **Contour element** menu item

▶ Select the contour element to be edited

▶ Present the selected contour element for shifting
▶ Enter the new **Start point** of the reference element

Over-
write

▶ Load the new **Start point** (= new position)
> The control displays the **shifted contour**.

Over-
write

▶ Confirm the new position for the contour

Transformations – Shifting

With this function, you can move a contour by entering incremental or absolute coordinates.

Parameters:

- **X**: **Target point**
- **Z**: **Target point**
- **Xi**: **Target point** (incremental)
- **Zi**: **Target point** (incremental)
- **H**: **Master** (only for C-axis contours)
  - **0: Delete**: Delete the master contour
  - **1: Copy**: Retain the master contour
- **ID**: **Contour** (only for C-axis contours)
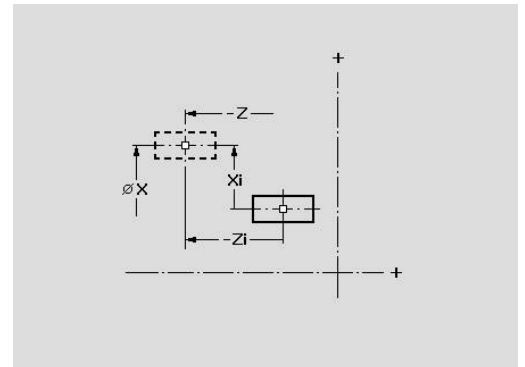
Transformations – Rotating

With this function you can rotate a contour about a center of rotation.

Parameters:

- **X**: **Center of rotation** (in Cartesian coordinates)
- **Z**: **Center of rotation** (in Cartesian coordinates)
- **W**: **Center of rotation** (in polar coordinates)
- **P**: **Center of rotation** (in polar coordinates)
- **A**: **Rotary angle**
- **H**: **Master** (only for C-axis contours)
  - **0: Delete**: Delete the master contour
  - **1: Copy**: Retain the master contour
- **ID**: **Contour** (only for C-axis contours)
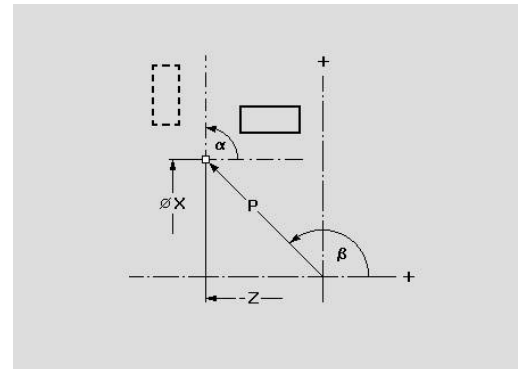
Soft keys
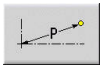
	Polar dimension of the center of rotation: Angle
	Polar dimension of the center of rotation: Radius

Transformations – Mirroring

This function mirrors the contour. Define the position of the **mirror axis** by entering the starting point and end point or the starting point and angle.

Parameters:

- **XS**: **Start point** (in Cartesian coordinates)
- **ZS**: **Start point** (in Cartesian coordinates)
- **X**: **Target point** (in Cartesian coordinates)
- **Z**: **Target point** (in Cartesian coordinates)
- **A**: **Angle** – Angle of rotation
- **WS**: **Start point** (in polar coordinates)
- **PS**: **Start point** (in polar coordinates)
- **W**: **Target point** (in polar coordinates)
- **P**: **Target point** (in polar coordinates)
- **H**: **Master** (only for C-axis contours)
  - **0: Delete**: Delete the master contour
  - **1: Copy**: Retain the master contour
- **ID**: **Contour** (only for C-axis contours)

Soft keys for polar dimensions

	Polar dimension of the starting point: Angle
	Polar dimension of the starting point: Radius
	Polar dimension of the end point: Angle
	Polar dimension of the end point: Radius
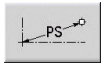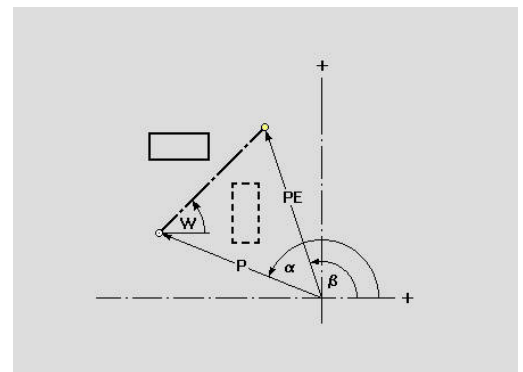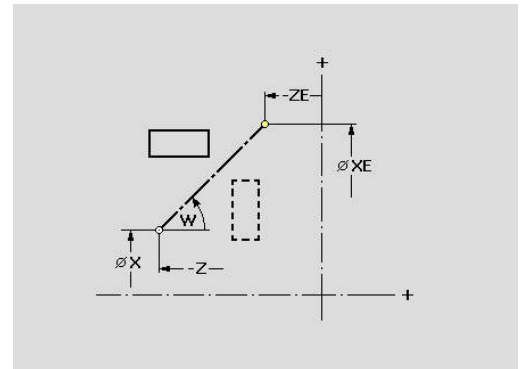
8.6 Zoom functions in ICP editor submode

The zoom functions make it possible to change the visible detail by using soft keys, the arrow keys, and the **PgUp** and **PgDn** keys. The **zoom** function can be called in all ICP windows.

The control sizes the detail depending on the programmed contour. With the zoom function you can select another detail.

Changing the detail

To modify the detail by using keys:

▶ The visible section can be modified with the arrow keys, and the **PgDn** and **PgUp** keys without opening the zoom menu

Keys for modifying the displayed detail

↓	↑	The arrow keys move the workpiece in the indicated direction
←	→	
PG DN		Magnify the displayed frame (zoom −)
PG UP		Reduce the displayed frame (zoom +)

To modify the detail with the zoom menu:

▶ If the zoom menu is selected, a red frame appears in the contour window. This red frame indicates the zoom area, which you can select using the **Apply** soft key or the **ENT** key. Use the following keys to change the frame size and position.
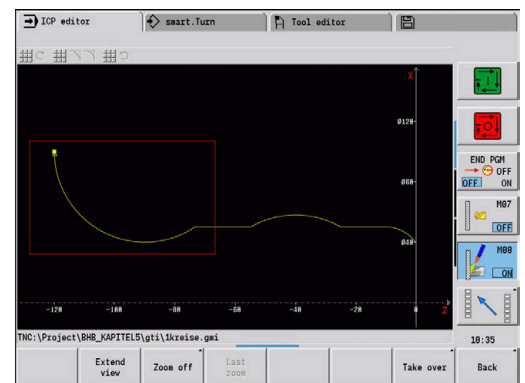
Keys for modifying the displayed detail

↓	↑	The arrow keys move the frame in the indicated direction
←	→	
PG DN		Reduce the displayed frame (zoom +)
PG UP		Magnify the displayed frame (zoom −)

Soft keys in the zoom menu

	Activate the zoom functions
Extend view	Magnify the displayed detail directly (zoom −)
Zoom off	Switch back to the standard screen detail and close the zoom menu
Last zoom	Return to the last screen detail selected
Take over	Load the area selected by the red frame as the new detail and close the zoom menu
Back	Close the zoom menu without changing the selected detail

8.7 Workpiece blank definition

In **smart.Turn** operating mode, the standard forms **Bar** and **Tube** are described with a G code.

Bar workpiece blank

The function describes a cylinder.

Parameters:

- **X**: **Diameter** of cylinder
- **Z**: **Length** of workpiece blank
- **K**: **Oversz** – Distance between workpiece datum and right edge

ICP generates a **G20** in the **BLANK** section in **smart.Turn** operating mode.
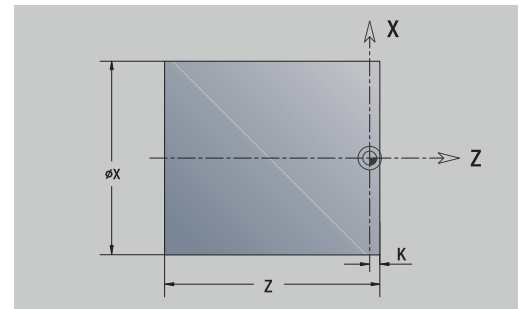
Tube workpiece blank

The function describes a hollow cylinder.

Parameters:

- **X**: **Outside diameter** – Diameter of hollow cylinder
- **I**: **Inner diameter (pipe)**
- **Z**: **Length** of workpiece blank
- **K**: **Oversz** – Distance between workpiece datum and right edge

ICP generates a **G20** in the **BLANK** section in **smart.Turn** mode of operation.
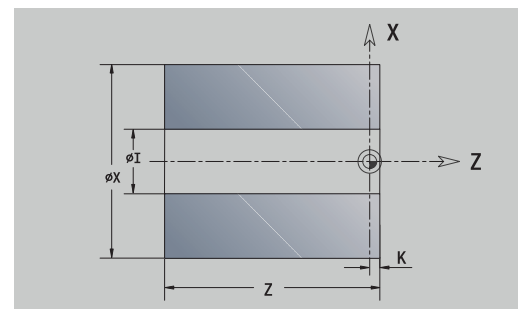
Cast part workpiece blank

The function describes an oversize on an existing finished part contour.

Parameters:

- **K**: **Contour-parallel oversize**

ICP generates a contour in the **BLANK** section in **smart.Turn** operating mode.
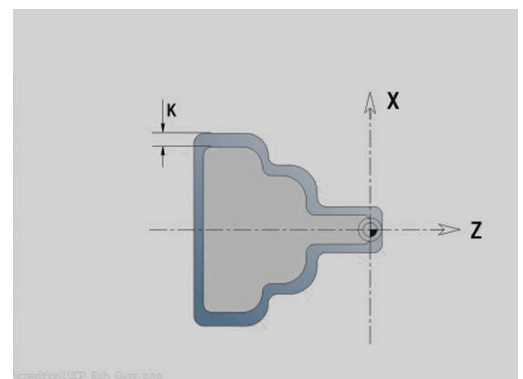
8.8 Contour elements – turning contour

With the contour elements of a turning contour you can create the
following:

- In **Teach-in** submode.
  - Complex workpiece blank contours
  - Contours for turning
- In **smart.Turn** operating mode
  - Complex workpiece blank contours and auxiliary workpiece
    blank contours
  - Finished part contours and auxiliary contours

Basic turning contour elements

Defining the starting point

Enter the coordinates for the starting point and target point in the
first contour element of the turning contour. Entering the starting
point is only possible in the first contour element. In subsequent
contour elements, the starting point results from the previous
contour element in each case.

To define the starting point:

▶ Select the **Contour** menu item
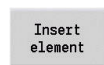
▶ Alternative: Press the **Insert element** soft key

Insert
element

▶ Select the contour element

Parameters for defining the starting point:

- **XS**, **ZS**: **Start point** of contour
- **WS**: **Starting point** of contour (angle in polar coordinates)
- **PS**: **Starting point** of contour (radius in polar coordinates)

ICP generates a **G0** in **smart.Turn** mode of operation.
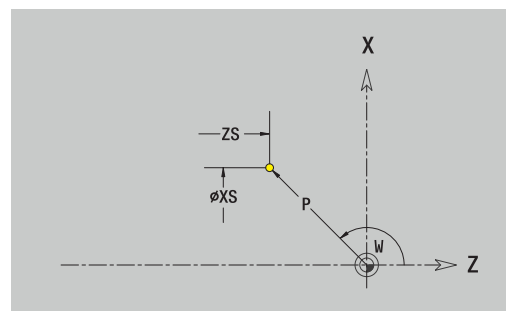
Vertical lines

To program a vertical line:

▶ Select the line direction

▶ Enter the line dimensions

▶ Define the transition to the next contour element

Parameters:

- **X**: **Target point**
- **Xi**: **Target point** (incremental)
- **W**: **Target point** (angle in polar coordinates)
- **P**: **Target point** (radius in polar coordinates)
- **L**: **Line length**
- **U**, **F**, **D**, **FP**, **IC**, **KC**, **HC**:
  Further information: "Machining attributes", Page 438

ICP generates a **G1** in **smart.Turn** mode of operation.

Horizontal lines

To program a horizontal line:

▶ Select the line direction

▶ Enter the line dimensions

▶ Define the transition to the next contour element

Parameters:

- **Z**: **Target point**
- **Zi**: **Target point** (incremental)
- **W**: **Target point** (angle in polar coordinates)
- **P**: **Target point** (radius in polar coordinates)
- **L**: **Line length**
- **U**, **F**, **D**, **FP**, **IC**, **KC**, **HC**:
  Further information: "Machining attributes", Page 438

ICP generates a **G1** in **smart.Turn** mode of operation.

Line at angle

To program a line at an angle:

▶ Select the line direction

▶ Enter the line dimensions
▶ Define the transition to the next contour element

Always enter the angle **AN** within the selected quadrant (<=90°).

Parameters:

- **X**, **Z**: **Target point**
- **Xi**, **Zi**: **Target point** (incremental)
- **W**: **Target point** (angle in polar coordinates)
- **P**: **Target point** (radius in polar coordinates)
- **L**: **Line length**
- **AN**: **Angle to Z axis**
- **ANn**: **Angle to Z axis** – Angle to the subsequent element
- **ANp**: **Angle to Z axis** – Angle to the previous element
- **U**, **F**, **D**, **FP**, **IC**, **KC**, **HC**:
  Further information: "Machining attributes", Page 438

ICP generates a **G1** in **smart.Turn** mode of operation.
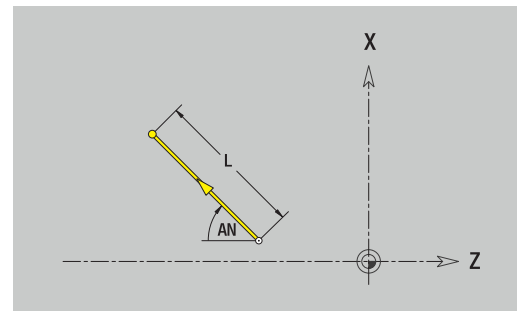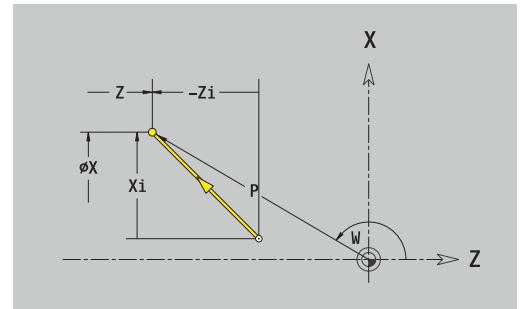
Circular arc

To program a circular arc:

▶ Select the arc's direction of rotation

▶ Enter the arc dimensions
▶ Define the transition to the next contour element

Parameters:
- **X**, **Z**: **Target point**
- **Xi**, **Zi**: **Target point** (incremental)
- **W**: **Target point** (angle in polar coordinates)
- **Wi**: **Target point** (polar angle, incremental; reference: starting point)
- **P**: **Target point** (radius in polar coordinates)
- **Pi**: **Target point** – Distance between starting point and target point (polar, incremental)
- **I**, **K**: **Center point** of circular arc
- **Ii**, **Ki**: **Center point** of circular arc (incremental) – Distance between starting point and center in **X** and **Z**
- **PM**: **Center point** of circular arc (radius in polar coordinates)
- **PMi**: **Center point** of circular arc – Distance between starting point and **Center point** (polar, incremental)
- **WM**: **Center point** of circular arc (polar angle)
- **WMi**: **Center point** of circular arc (polar angle, incremental; reference: starting point)
- **R**: **Radius**
- **ANs**: **Angle** – Tangential angle in the starting point
- **ANe**: **Angle** – Tangential angle at the target point
- **ANn**: **Angle** to the subsequent element
- **ANp**: **Angle** to the previous element
- **U**, **F**, **D**, **FP**:
  Further information: "Machining attributes", Page 438

ICP generates a **G2** or **G3** in **smart.Turn** mode of operation.
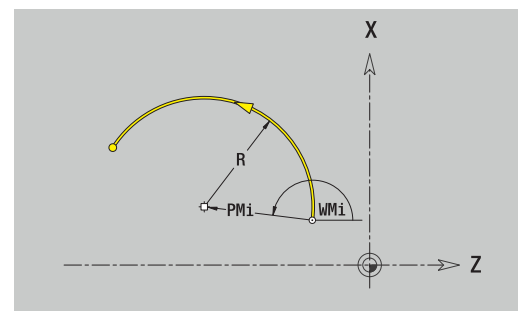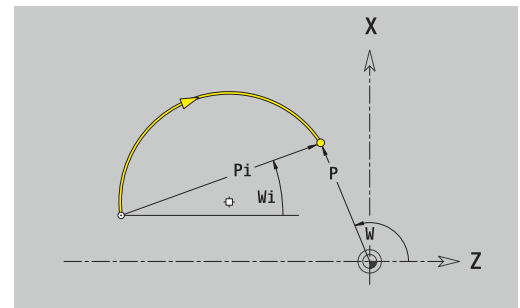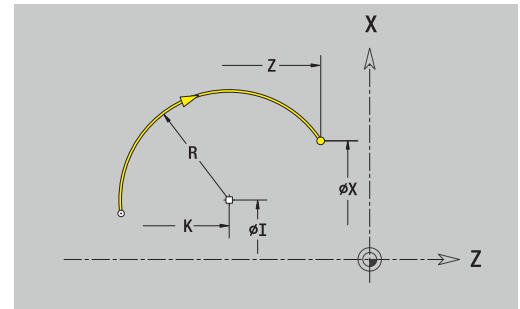
Turning contour form elements

Chamfer or rounding

To program a chamfer or rounding radius:

▶ Select the form element menu

▶ Select the Chamfer menu item

▶ Enter the **Cham. width BR**

▶ Alternative: Select the Rounding menu item

▶ Enter the **Rounding radius BR**

▶ Enter the chamfer or rounding as first contour element: **Element pos AN**

Parameters:

- **BR**: **Cham. width** or **Rounding radius**
- **AN**: **Element pos**
- **U**, **F**, **D**, **FP**:
  Further information: "Machining attributes", Page 438

Chamfers or roundings are defined at contour corners. A **contour corner** is the point of intersection between the approaching contour element and the departing contour element. A chamfer or rounding cannot be calculated until the basic contour element is known.

ICP integrates the chamfer or rounding in the **G1**, **G2** or **G3** basic element in **smart.Turn** mode of operation.

Contour begins with a chamfer or a rounding: Enter the position of the **intended** corner as starting point. Then, in the form element menu, select Chamfer or Rounding. Since the **approaching contour element** is missing, you enter the **element position AN** to clearly define the position of the chamfer or rounding radius.

Example of an outside chamfer at start of contour: If you program **Element pos AN** = 90°, the intended approaching reference element is a planar element in the positive **+ X direction**.

ICP converts a chamfer or rounding at the start of the contour to a linear or circular element.
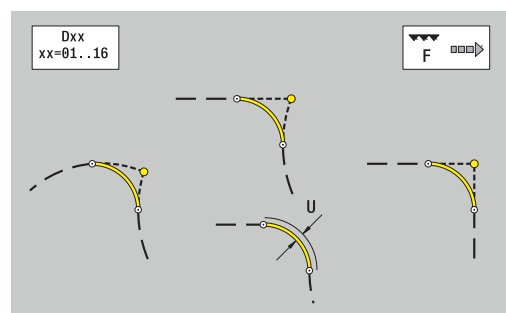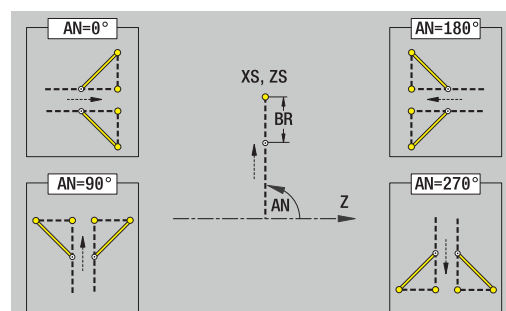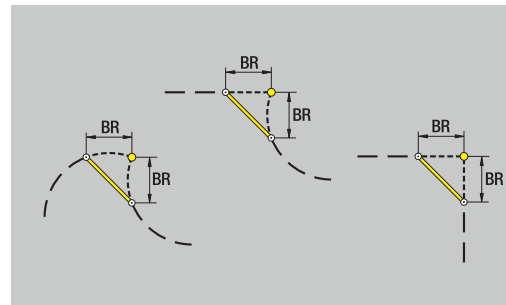
Undercut DIN 76

To program a thread undercut DIN 76:

▶ Select the form element menu

▶ Select **Undercut DIN 76**

▶ Enter the undercut parameters

Parameters:

- **FP**: **Thread pitch** (default: value from standard table)
- **I**: **Undercut depth** (default: value from standard table)
- **K**: **Undercut length** (default: standard table)
- **R**: **Undercut radius** (default: standard table)
- **W**: **Undercut angle** (default: value from standard table)
- **U**, **F**, **D**, **DF**:
  Further information: "Machining attributes", Page 438

ICP generates a **G25** in **smart.Turn** mode of operation.

Parameters that are not defined are automatically calculated by the control from the standard table:

- The **Thread pitch FP** is calculated from the diameter
- The parameters **I**, **K**, **W**, and **R** are calculated from the **Thread pitch FP**

Further information: "DIN 76 – undercut parameters", Page 779

> ℹ️ - If you are programming an internal thread, it is advisable to preset the **Thread pitch FP**, since the diameter of the longitudinal element is not the thread diameter. If you have the control calculate the thread pitch automatically, slight deviations may occur.
> - Undercuts can be programmed only between two linear elements. One of the two linear elements must be parallel to the X axis.
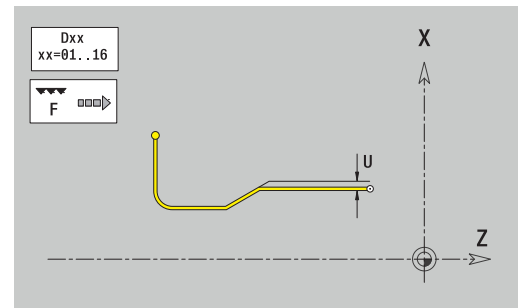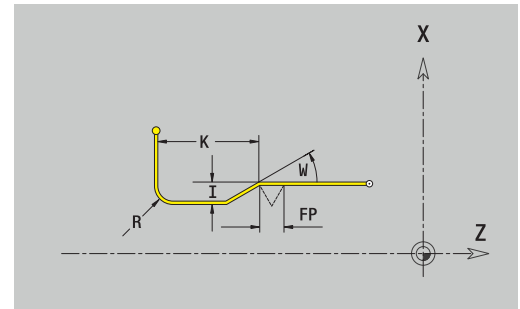
Undercut DIN 509 E

To program a DIN 509 E undercut:

▶ Select the form element menu

▶ Select **Undercut DIN 509 E**

▶ Enter the undercut parameters

Parameters:

- **I**: **Undercut depth** (default: value from standard table)
- **K**: **Undercut length** (default: standard table)
- **R**: **Undercut radius** (default: standard table)
- **W**: **Undercut angle** (default: value from standard table)
- **U**, **F**, **D**, **DF**:
  Further information: "Machining attributes", Page 438

ICP generates a **G25** in **smart.Turn** mode of operation.

The control automatically calculates parameters that have not been entered from the diameter in the standard table.

Further information: "DIN 509 E – undercut parameters", Page 780

 Undercuts can be programmed only between two linear elements. One of the two linear elements must be parallel to the X axis.

Undercut DIN 509 F

To program a DIN 509 F undercut:

▶ Select the form element menu

▶ Select **Undercut DIN 509 F**

▶ Enter the undercut parameters

Parameters:

- **I**: **Undercut depth** (default: value from standard table)
- **K**: **Undercut length** (default: standard table)
- **R**: **Undercut radius** (default: standard table)
- **W**: **Undercut angle** (default: value from standard table)
- **P**: **Trav.dpth** (default: value from standard table)
- **A**: **Face angle** (default: standard table)
- **U**, **F**, **D**, **DF**:
  Further information: "Machining attributes", Page 438

ICP generates a **G25** in **smart.Turn** mode of operation.

The control automatically calculates parameters that have not been entered from the diameter in the standard table.

Further information: "DIN 509 F – undercut parameters", Page 780
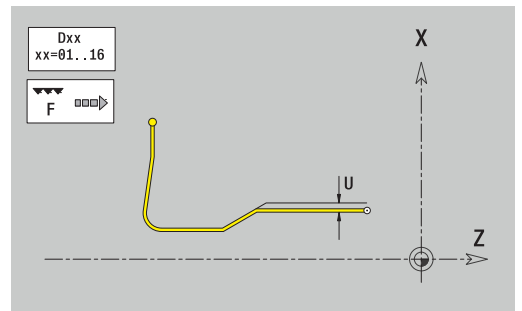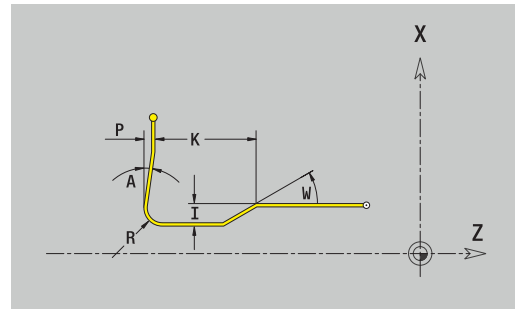
> **ℹ** Undercuts can be programmed only between two linear elements. One of the two linear elements must be parallel to the X axis.

Undercut form U

To program an undercut type U:

▶ Select the form element menu

▶ Select **Undercut form U**

▶ Enter the undercut parameters

Parameters:

- **I**: **Undercut depth**
- **K**: **Length**
- **R**: **Undercut radius**
- **P**: **Chamf./round.**
- **U**, **F**, **D**, **DF**:
  Further information: "Machining attributes", Page 438

ICP generates a **G25** in **smart.Turn** mode of operation.
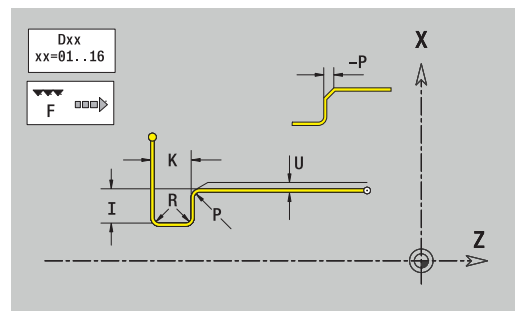
> **ℹ** Undercuts can be programmed only between two linear elements. One of the two linear elements must be parallel to the X axis.

Undercut form H

To program an undercut type H:

▶ Select the form element menu

▶ Select **Undercut form H**

▶ Enter the undercut parameters

Parameters:
- **K**: **Length**
- **R**: **Undercut radius**
- **W**: **Plunging angle**
- **U**, **F**, **D**, **DF**:
  Further information: "Machining attributes", Page 438

ICP generates a **G25** in **smart.Turn** mode of operation.
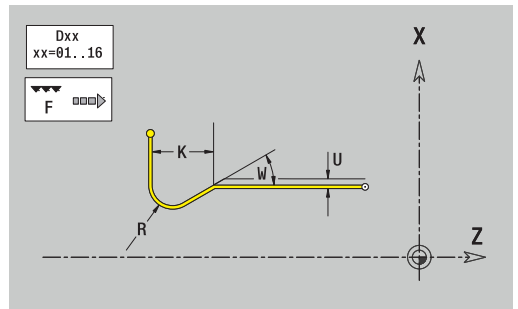
> 🛈 Undercuts can be programmed only between two linear elements. One of the two linear elements must be parallel to the X axis.

Undercut form K

To program an undercut type K:

▶ Select the form element menu

▶ Select **Undercut form K**

▶ Enter the undercut parameters

Parameters:
- **I**: **Undercut depth**
- **R**: **Undercut radius**
- **W**: **Opening angle**
- **A**: **Plunging angle**
- **U**, **F**, **D**, **DF**:
  Further information: "Machining attributes", Page 438

ICP generates a **G25** in **smart.Turn** mode of operation.
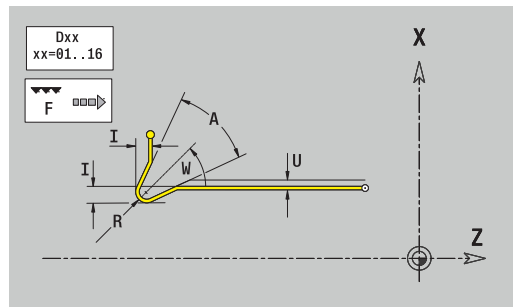
> 🛈 Undercuts can be programmed only between two linear elements. One of the two linear elements must be parallel to the X axis.

8.9 Contour elements – front face

With the contour elements of a face you can create complex milling contours.

- In **Teach-in** submode: Contours for axial ICP milling cycles
- In **smart.Turn** mode of operation: Contours for machining with the C axis

Enter the dimensions of the front face contour elements in Cartesian or polar coordinates. You can switch between them by soft key. You can mix Cartesian coordinates and polar coordinates to define a point.

Soft keys for polar coordinates

	Switch the field to entering the angle **C**
	Switch the field to entering the radius **P**

Basic elements of face

Starting point of face contour

Enter the coordinates for the starting point and target point in the first contour element. Entering the starting point is only possible in the first contour element. In subsequent contour elements, the starting point results from the previous contour element in each case.

To define the starting point:

▶ Select the **Contour** menu item
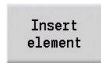
▶ Alternative: Press the **Insert element** soft key

▶ Define the starting point

Parameters for defining the starting point:

- **XKS**, **YKS**: **Starting point** of contour
- **CS**: **Starting point** of contour (angle in polar coordinates)
- **PS**: **Starting point** of contour (radius in polar coordinates)
- **HC**: **Milling/drilling attribute**
    - **1: Contour milling G840/G847**
    - **2: Pocket milling G845/G848**
    - **3: Area milling G841-G844**
    - **4: Deburring G840**
    - **5: Engraving G801-G804**
    - **6: Contour + deburring G840/G847**
    - **7: Pocket + deburring G845/G848**
    - **8: face milling G797**
    - **9: face milling + deburring G797**
    - **10: bore milling G75**
    - **11: thread milling G799/G800/G806**
    - **12: bore and thread milling G75/G799..**
    - **14: Do not machine**
- **QF**: **Cutter position**
    - **0: On the contour**
    - **1: Inside / left**
    - **2: Outside / right**
- **HF**: **Direction**
    - **0: Up-cut**
    - **1: Climb**
- **DF**: **Milling diameter**
- **WF**: **Angle** of chamfer
- **BR**: **Chamfer width**
- **RB**: **Return plane**

ICP generates a **G100** in **smart.Turn** mode of operation.
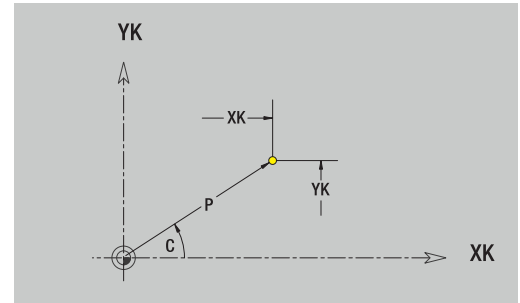
Vertical lines on front face

To program vertical lines:
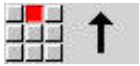
▶ Select the line direction

▶ Enter the line dimensions

▶ Define the transition to the next contour element

Parameters:

- **YK**: **Target point** (in Cartesian coordinates)
- **YKi**: **Target point** (incremental) – Distance between starting point and **Target point**
- **C**: **Target point** (angle in polar coordinates)
- **P**: **Target point** (in polar coordinates)
- **L**: **Line length**
- **F**: **Further information:** "Machining attributes", Page 438

ICP generates a **G101** in **smart.Turn** mode of operation.

Horizontal lines on front face
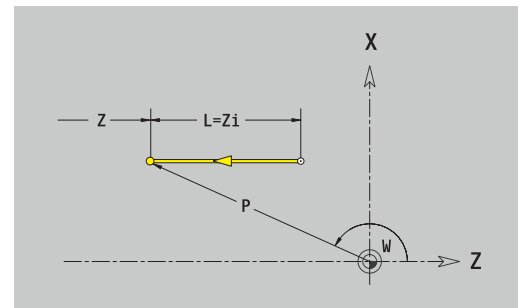
To program horizontal lines:
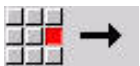
▶ Select the line direction

▶ Enter the line dimensions

▶ Define the transition to the next contour element

Parameters:

- **XK**: **Target point** (in Cartesian coordinates)
- **XKi**: **Target point** (incremental) – Distance between starting point and **Target point**
- **C**: **Target point** (angle in polar coordinates)
- **P**: **Target point** (in polar coordinates)
- **L**: **Line length**
- **F**: **Further information:** "Machining attributes", Page 438

ICP generates a **G101** in **smart.Turn** mode of operation.
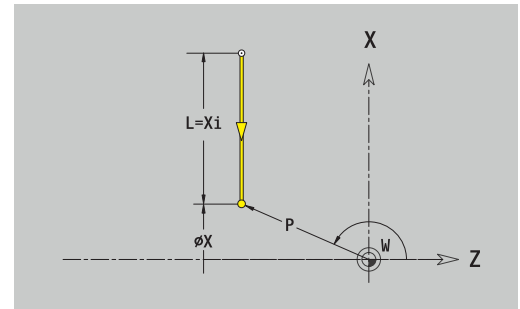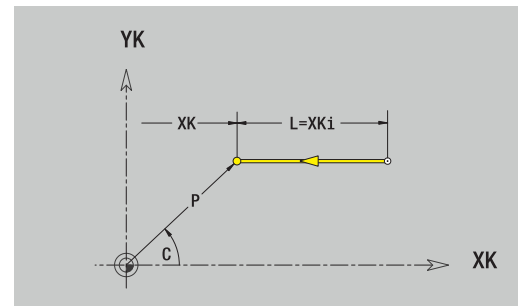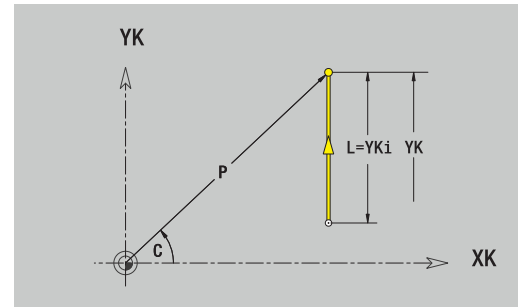
Line at angle on front face

Programming a line at an angle:
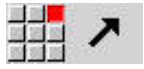
▶ Select the line direction

▶ Enter the line dimensions

▶ Define the transition to the next contour element

Parameters:

- **XK**, **YK**: **Target point** (in Cartesian coordinates)
- **XKi**, **YKi**: **Target point** (incremental) – Distance between starting point and **Target point**
- **C**: **Target point** (angle in polar coordinates)
- **P**: **Target point** (in polar coordinates)
- **L**: **Line length**
- **AN**: **Angle** to positive XK axis
- **ANn**: **Angle** to the subsequent element
- **ANp**: **Angle** to the previous element
- **F**: **Further information:** "Machining attributes", Page 438

ICP generates a **G101** in **smart.Turn** mode of operation.
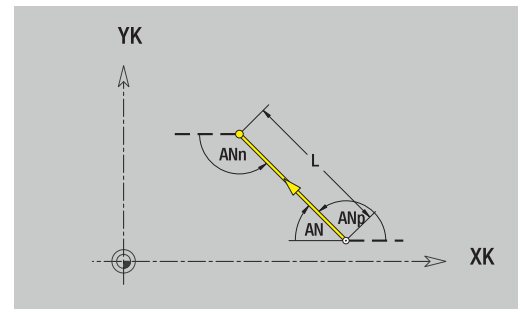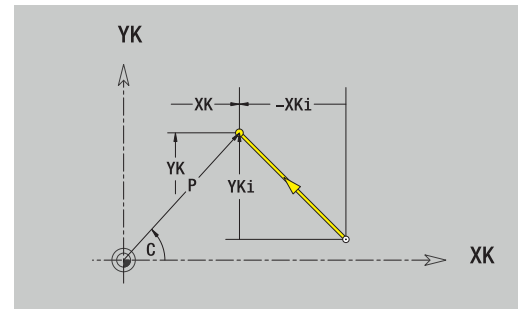
Circular arc on front face

To program a circular arc:

▶ Select the arc's direction of rotation

▶ Enter the arc dimensions

▶ Define the transition to the next contour element

Parameters:

- **XK**, **YK**: **Target point** – End point of circular arc
- **XKi**, **YKi**: **Target point** (incremental) – Distance between starting point and **Target point**
- **P**: **Target point** (in polar coordinates)
- **Pi**: **Target point** – Distance between starting point and target point (polar, incremental)
- **C**: **Target point** (angle in polar coordinates)
- **Ci**: **Target point** (polar angle, incremental; reference: starting point)
- **I**, **J**: **Center point** of circular arc
- **Ii**, **Ji**: **Center point** of circular arc (incremental) – Distance between starting point and **Center point** in **X** and **Z**
- **PM**: **Center point** of circular arc (in polar coordinates)
- **PMi**: **Center point** of circular arc – Distance between starting point and **Center point** (polar, incremental)
- **CM**: **Center point** of circular arc (polar angle)
- **CMi**: **Center point** of circular arc (polar angle, incremental; reference: starting point)
- **R**: **Radius**
- **ANs**: **Angle** – Tangential angle in the starting point
- **ANe**: **Angle** – Tangential angle at the target point
- **ANn**: **Angle** to the subsequent element
- **ANp**: **Angle** to the previous element
- **F**: **Further information:** "Machining attributes", Page 438

ICP generates a **G102** or **G103** in **smart.Turn** mode of operation.
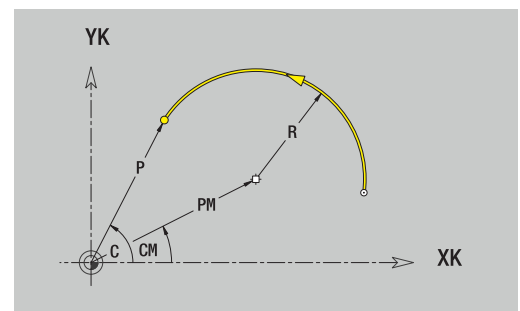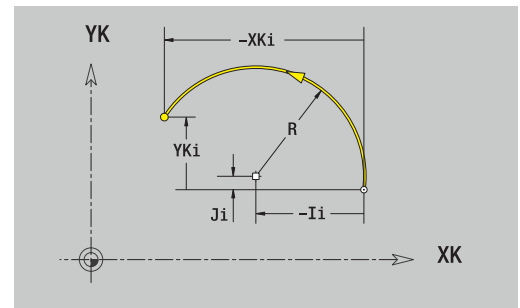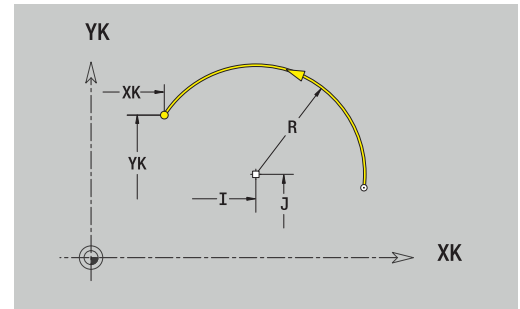
Form elements of face

Chamfer or rounding on front face

To program a chamfer or rounding radius:

▶ Select the form element menu

▶ Select the Chamfer menu item

▶ Enter the **Cham. width BR**

▶ Alternative: Select the Rounding menu item

▶ Enter the **Rounding radius BR**

▶ Enter the chamfer or rounding as first contour element: **Element pos AN**

Parameters:

- **BR**: **Cham. width** or **Rounding radius**
- **AN**: **Element pos**
- **F**: **Further information:** "Machining attributes", Page 438

Chamfers or roundings are defined at contour corners. A **contour corner** is the point of intersection between the approaching contour element and the departing contour element. A chamfer or rounding cannot be calculated until the basic contour element is known.

ICP integrates the chamfer or rounding in the **G101**, **G102** or **G103** basic element in **smart.Turn** mode of operation.

Contour begins with a chamfer or a rounding: Enter the position of the **intended** corner as starting point. Then, in the form element menu, select Chamfer or Rounding. Since the **approaching contour element** is missing, you enter the **Element pos AN** to clearly define the position of the chamfer or rounding.

Example of an outside chamfer at start of contour: If you program **Element pos AN** = 90°, the intended approaching reference element is a transverse element in the positive **+ X direction**.

ICP converts a chamfer or rounding at the start of the contour to a linear or circular element.
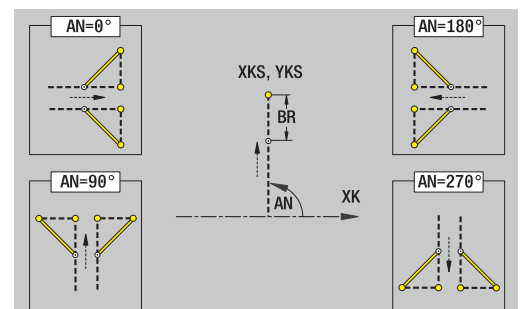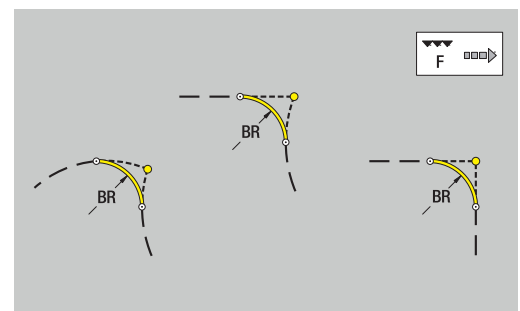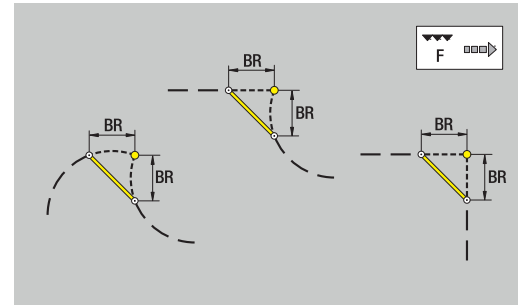
8.10 Contour elements on lateral surface

With the contour elements of a lateral surface you can create complex milling contours.

- In **Teach-in** submode: Contours for radial ICP milling cycles
- In **smart.Turn** mode of operation: Contours for machining with the C axis

Enter the dimensions of the lateral surface contour elements in Cartesian or polar coordinates. You can use the linear dimension as an alternative to the angular dimension. You can switch between them by soft key.

> **i** The linear dimension is essentially the unrolled surface at the reference diameter.
>
> - For lateral surface contours, the reference diameter is specified in the cycle. The linear dimensions of all subsequent contour elements are referenced to this diameter
> - When called from **smart.Turn** mode of operation, the reference diameter in the reference data is requested

---

Soft keys for polar coordinates

	Switch the field from linear dimensions to entry of the angle **C**
	Switch the field to entering the polar dimension **P**

Basic elements of lateral surface

Starting point of lateral surface contour

Enter the coordinates for the starting point and target point in the first contour element. Entering the starting point is only possible in the first contour element. In subsequent contour elements, the starting point results from the previous contour element in each case.

To define the starting point:

▶ Select the **Contour** menu item

▶ Alternative: Press the **Insert element** soft key

▶ Define the starting point

Parameters for defining the starting point:

- **ZS**: **Starting point** of contour
- **CYS**: **Starting point** of contour as linear dimension (reference: diameter **XS**)
- **PS**: **Starting point** of contour (radius in polar coordinates)
- **PS**: **Starting point** of contour (in polar coordinates)
- **HC**: **Milling/drilling attribute**
    - **1: Contour milling G840/G847**
    - **2: Pocket milling G845/G848**
    - **3: Area milling G841-G844**
    - **4: Deburring G840**
    - **5: Engraving G801-G804**
    - **6: Contour + deburring G840/G847**
    - **7: Pocket + deburring G845/G848**
    - **8: face milling G797**
    - **9: face milling + deburring G797**
    - **10: bore milling G75**
    - **11: thread milling G799/G800/G806**
    - **12: bore and thread milling G75/G799..**
    - **14: Do not machine**
- **QF**: **Cutter position**
    - **0: On the contour**
    - **1: Inside / left**
    - **2: Outside / right**
- **HF**: **Direction**
    - **0: Up-cut**
    - **1: Climb**
- **DF**: **Milling diameter**
- **WF**: **Angle** of chamfer
- **BR**: **Chamfer width**
- **RB**: **Return plane**

ICP generates a **G110** in **smart.Turn** mode of operation.
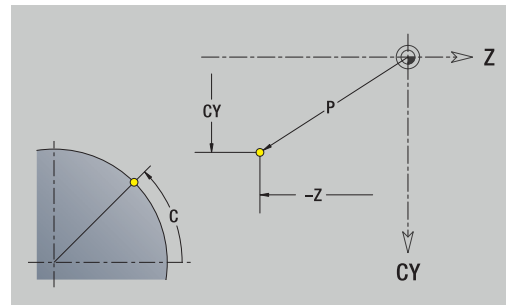
Vertical lines on lateral surface

To program vertical lines:

- ▶ Select the line direction
- ▶ Enter the line dimensions
- ▶ Define the transition to the next contour element

Parameters:

- **CY**: **Target point** as linear dimension (reference: diameter **XS**)
- **CYi**: **Target point** (incremental) as linear dimension (reference: diameter **XS**)
- **C**: **Target point** (angle in polar coordinates)
- **P**: **Target point** (in polar coordinates)
- **L**: **Line length**
- **F**: **Further information:** "Machining attributes", Page 438

ICP generates a **G111** in **smart.Turn** mode of operation.

Horizontal lines on lateral surface

To program horizontal lines:

- ▶ Select the line direction
- ▶ Enter the line dimensions
- ▶ Define the transition to the next contour element

Parameters:

- **Z**: **Target point**
- **Zi**: **Target point** (incremental)
- **P**: **Target point** (in polar coordinates)
- **L**: **Line length**
- **F**: **Further information:** "Machining attributes", Page 438

ICP generates a **G111** in **smart.Turn** mode of operation.
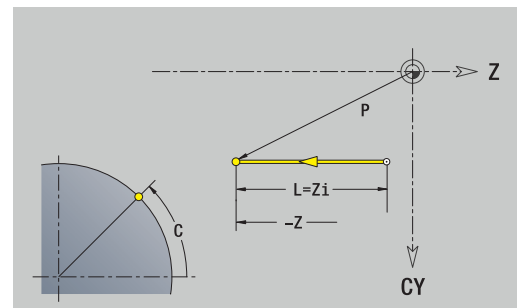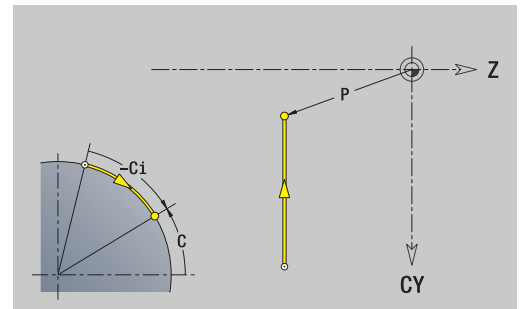
Line at angle on lateral surface

To program a line at an angle:
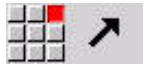
▶ Select the line direction

▶ Enter the line dimensions

▶ Define the transition to the next contour element

Parameters:

- **Z**: **Target point**
- **Zi**: **Target point** (incremental)
- **CY**: **Target point** as linear dimension (reference: diameter **XS**)
- **CYi**: **Target point** (incremental) as linear dimension (reference: diameter **XS**)
- **P**: **Target point** (in polar coordinates)
- **C**: **Target point** (angle in polar coordinates)
- **Ci**: **Target point** (polar angle, incremental)
- **AN**: **Angle** to the Z axis (for direction of angle, see help graphics)
- **ANn**: **Angle** to the subsequent element
- **ANp**: **Angle** to the previous element
- **L**: **Line length**
- **F**: **Further information:** "Machining attributes", Page 438

ICP generates a **G111** in **smart.Turn** mode of operation.

Circular arc on lateral surface

To program a circular arc:

▶ Select the arc's direction of rotation

▶ Enter the arc dimensions

▶ Define the transition to the next contour element

Parameters:

- **Z**: **Target point**
- **Zi**: **Target point** (incremental)
- **CY**: **Target point** as linear dimension (reference: diameter **XS**)
- **CYi**: **Target point** (incremental) as linear dimension (reference: diameter **XS**)
- **P**: **Target point** (in polar coordinates)
- **Pi**: **Target point** – Distance between starting point and target point (polar, incremental)
- **C**: **Target point** (angle in polar coordinates)
- **Ci**: **Target point** (polar angle, incremental)
- **K**: **Center point** in **Z**
- **Ki**: **Center point** (incremental; in **Z**)
- **CJ**: **Center point** as linear dimension (reference: diameter **XS**)
- **CJi**: **Center point** (incremental) as linear dimension (reference: diameter **XS**)
- **PM**: **Center point** of circular arc (in polar coordinates)
- **PMi**: **Center point** of circular arc – Distance between starting point and **Center point** (polar, incremental)
- **WM**: **Center point** of circular arc (polar angle)
- **WMi**: **Center point** of circular arc (polar angle, incremental; reference: starting point)
- **R**: **Radius**
- **ANs**: **Angle** – Tangential angle in the starting point
- **ANe**: **Angle** – Tangential angle at the target point
- **ANn**: **Angle** to the subsequent element
- **ANp**: **Angle** to the previous element
- **L**: **Line length**
- **F**: **Further information:** "Machining attributes", Page 438

ICP generates a **G112** or **G113** in **smart.Turn** mode of operation.
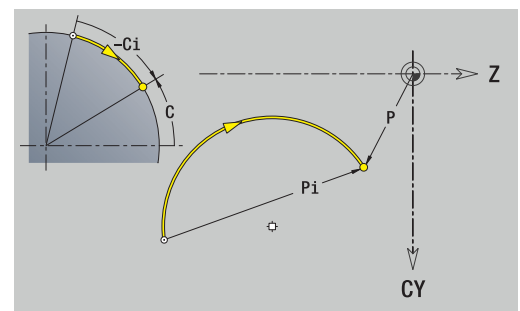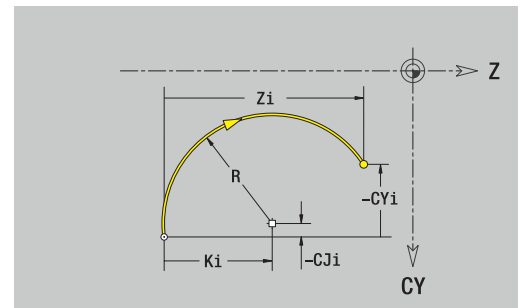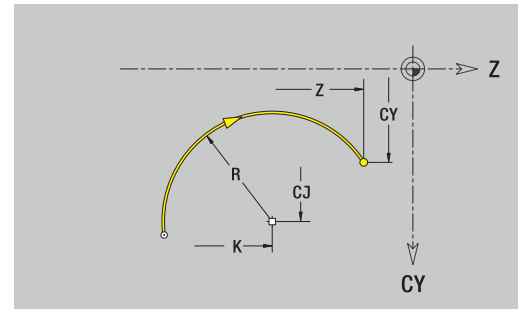
Form elements of lateral surface

Chamfer or rounding on lateral surface

To program a chamfer or rounding radius:

▶ Select the form element menu

▶ Select the Chamfer menu item

▶ Enter the **Cham. width BR**

▶ Alternative: Select the Rounding menu item

▶ Enter the **Rounding radius BR**

▶ Enter the chamfer or rounding as first contour element: **Element pos AN**

Parameters:

- **BR**: **Cham. width** or **Rounding radius**
- **AN**: **Element pos**
- **F**: **Further information:** "Machining attributes", Page 438

Chamfers or roundings are defined at contour corners. A **contour corner** is the point of intersection between the approaching contour element and the departing contour element. A chamfer or rounding cannot be calculated until the basic contour element is known.

ICP integrates the chamfer or rounding in the **G111**, **G112** or **G113** basic element in **smart.Turn** mode of operation.

Contour begins with a chamfer or a rounding: Enter the position of the **intended** corner as starting point. Then, in the form element menu, select Chamfer or Rounding. Since the **approaching contour element** is missing, you enter the **Element pos AN** to clearly define the position of the chamfer or rounding.

ICP converts a chamfer or rounding at the start of the contour to a linear or circular element.
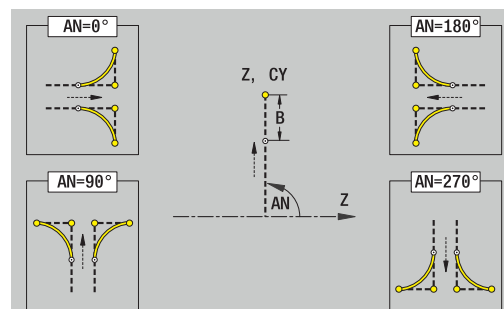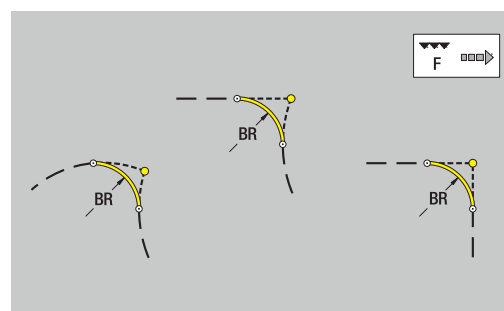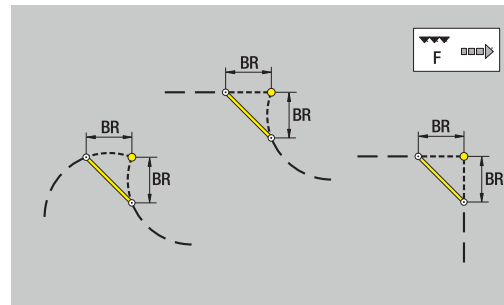
8.11 C-axis and Y-axis machining in smart.Turn operating mode (option 9)

In **smart.Turn** mode of operation, ICP supports the definition of milling contours and holes as well as the creation of milling and drilling patterns that are machined with the aid of the C axis or Y axis.

Before you use ICP to describe a milling contour or a hole, select the plane:

- C axis
    - Frontface (XC plane)
    - Surface (ZC plane)
- Y axis
    - Frontface (XY plane)
    - Surface (YZ plane)

A hole can contain the following elements:

- Centering
- Core hole
- Cavity
- Thread

The parameters are evaluated for drilling or tapping.

You can arrange holes in linear or circular patterns.

Milling contours: The control is familiar with standard figures (full circle, polygon, slot, etc.). You can define these figures with a few parameters. Complex contours are described with lines and circular arcs.

You can arrange standard figures in linear or circular patterns.

Reference data and nested contours

When describing a milling contour or hole, you define the **reference plane**. The reference plane is the position on which the milling contour or the hole is created.

- Frontface (C axis): The Z position (Reference dimension)
- Surface (C axis): The X position (Reference diameter)
- Frontface (Y axis): The Z position (Reference dimension)
- Surface (Y axis): The X position (Reference diameter)

It is also possible to **nest** milling contours and holes. Example: Defining a slot in a rectangular pocket. Holes are drilled inside this slot. You use the reference plane to specify the positions of these elements.

ICP supports the selection of the reference plane. The following reference data are loaded during selection of a reference plane.

- Frontface: Reference dimension
- Surface: Reference diameter
- Frontface: Reference dimension, spindle angle, limit diameter
- Surface: Reference diameter, spindle angle

To select a reference plane:

- ▶ Select contour, figure, hole, pattern, single surface or polygon

Select reference plane	▶ Press the **Select reference plane** soft key > **ICP** displays the finished part and, if available, the contours already defined ▶ Use the soft keys (see table) to select the reference dimension, reference diameter or existing milling contour as reference plane
Select	▶ Confirm the reference plane > **ICP** loads the values of the reference plane as reference data ▶ Complete the reference data and describe the contour, figure, hole, pattern, single surface or polygon

Soft keys for nested contours

	Switch to the next contour of the same reference plane
	Switch to the previous contour of the same reference plane
	Switch to the next nested contour
	Switch to the previous nested contour

Representation of the ICP elements in the smart.Turn program

Each ICP dialog in smart.Turn programs is represented by a section code followed by further **G** codes.

A hole or milling contour (standard figure and complex contour) contains the following commands:

- Section code (with the reference data of this section):
  - **FRONT** (XC plane)
  - **LATERAL** (ZC plane)
  - **FRONT Y** (XY plane)
  - **SURFACE Y** (ZY plane)
- **G308** (with parameters) as beginning of the reference plane
- **G** code of the figure or hole; sequence of commands for patterns or complex contours
- **G309** as end of a reference plane

In nested contours a reference plane begins with **G308**, the next reference plane with the next **G308**, etc. When the **deepest level** is finally reached, this reference plane is concluded with **G309**. Then the next reference plane is concluded with **G309**, etc.

Note the following points if you describe milling contours or holes with **G** codes and then edit them with **ICP**:

- Some parameters are redundant in the DIN contour description. For example, the milling depth can be programmed in the **G308** and/or in the **G** code of the figure. This redundancy does not appear in **ICP**.
- In DIN programming of figures you can choose between Cartesian or polar center coordinates. In **ICP**, the center of the figures is entered in Cartesian coordinates.

Example

In the DIN contour description, the milling depth is programmed in **G308** and in the figure definition. If you change this figure using **ICP**, **ICP** overwrites the milling depth from **G308** with the milling depth from the figure. When saving, **ICP** saves the milling depth in **G308**. The **G** code of the figure is saved without milling depth.

Example: Rectangle on face

...
FRONT Z0
N 100 G308 ID"STIRN_1" P-5
N 101   G305 XK40 YK10 A0 K30 B15
N 102 G309
...

Example: Nested figures

...
FRONT Z0
N 100 G308 ID"STIRN_2" P-5
N 101   G307 XK-40 YK-40 Q5 A0 K-50
N 102   G308 ID"STIRN_12" P-3
N 103   G301 XK-35 YK-40 A30 K40 B20
N 104   G309
N 105 G309
...

8.12 Front face contours in smart.Turn operating mode (option 9)

In **smart.Turn** mode of operation, **ICP** provides the following contours for machining with the C axis:

- Complex contours defined with individual contour elements
- Figures
- Holes
- Pattern of figures or holes

Reference data for complex face contours

The reference data is followed by the contour definition with individual contour elements:

Further information: "Contour elements – front face", Page 477

Reference data of face:

- **ID**: **Contour**
- **PT**: **Milling depth**
- **ZR**: **Reference dimension**

You can use the **Select reference plane** function to determine the **Reference dimension ZR**.

Further information: "Reference data and nested contours", Page 490

ICP generates the following:

- The **FRONT** section code with the **Reference dimension** parameter. In nested contours, **ICP** generates only one section code
- A **G308** with the parameters **Contour name** and **Milling depth**
- A **G309** at the end of the contour description

TURN PLUS attributes

In the TURN PLUS attributes, you can define settings for the **Automatic working plan generation** (**AWG**) submode.

Parameters for defining the starting point:

- **HC**: **Milling/drilling attribute**
    - **1: Contour milling G840/G847**
    - **2: Pocket milling G845/G848**
    - **3: Area milling G841-G844**
    - **4: Deburring G840**
    - **5: Engraving G801-G804**
    - **6: Contour + deburring G840/G847**
    - **7: Pocket + deburring G845/G848**
    - **8: face milling G797**
    - **9: face milling + deburring G797**
    - **10: bore milling G75**
    - **11: thread milling G799/G800/G806**
    - **12: bore and thread milling G75/G799..**
    - **14: Do not machine**
- **DF**: **Milling diameter**
- **QF**: **Cutter position**
    - **0: On the contour**
    - **1: Inside / left**
    - **2: Outside / right**
- **HF**: **Direction**
    - **0: Up-cut**
    - **1: Climb**
- **OF**: **Plunging behavior**
    - **0 / no input – vertical plunging**
    - **1: Helical plunging**
        - In roughing cycles, the tool plunges in a reciprocating movement when milling pockets, otherwise—i.e. when milling slots—in a helical movement.
        - In finishing cycles, the tool plunges in a 3-D approach arc when milling pockets.
    - **2: Reciprocating plunge**
        - In roughing cycles, the tool plunges in a reciprocating movement when milling pockets.
        - In finishing cycles, the tool plunges in a 3-D approach arc when milling pockets.
- **IF**: **Limit diameter**
- **RC**: **Trochoid width**
- **RB**: **Return plane**
- **BF**: **Chamfer width**
- **WF**: **Angle** of chamfer

Circle on front face

Reference data of face:

- **ID**: **Contour**
- **PT**: **Milling depth**
- **ZR**: **Reference dimension**

Parameters of figure:

- **XKM**, **YKM**: **Center point** of figure (in Cartesian coordinates)
- **MPX**: **Diameter** – **Center** (in polar coordinates)
- **MPC**: **Angle** – **Center** (in polar coordinates)
- **R**: **Radius**

You can use the **Select reference plane** function to determine the **Reference dimension ZR**.

Further information: "Reference data and nested contours", Page 490

ICP generates the following:

- The **FRONT** section code with the **Reference dimension** parameter. In nested contours, **ICP** generates only one section code
- A **G308** with the parameters **Contour name** and **Milling depth**
- A **G304** with the parameters of the figure
- A **G309**

Rectangle on front face

Reference data of face:

- **ID**: **Contour**
- **PT**: **Milling depth**
- **ZR**: **Reference dimension**

Parameters of figure:

- **XKM**, **YKM**: **Center point** of figure (in Cartesian coordinates)
- **MPX**: **Diameter** – **Center** (in polar coordinates)
- **MPC**: **Angle** – **Center** (in polar coordinates)
- **A**: **Position angle** (reference: XK axis)
- **K**: **Length**
- **B**: **Width**
- **BR**: **Cham. width** or **Rounding radius**

You can use the **Select reference plane** function to determine the **Reference dimension ZR**.

Further information: "Reference data and nested contours", Page 490

ICP generates the following:

- The **FRONT** section code with the **Reference dimension** parameter. In nested contours, **ICP** generates only one section code
- A **G308** with the parameters **Contour name** and **Milling depth**
- A **G305** with the parameters of the figure
- A **G309**
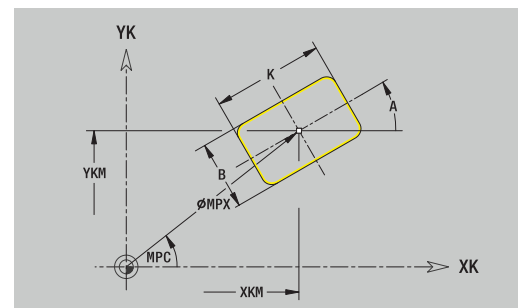
Polygon on front face

Reference data of face:

- **ID**: **Contour**
- **PT**: **Milling depth**
- **ZR**: **Reference dimension**

Parameters of figure:

- **XKM**, **YKM**: **Center point** of figure (in Cartesian coordinates)
- **MPX**: **Diameter** – **Center** (in polar coordinates)
- **MPC**: **Angle** – **Center** (in polar coordinates)
- **A**: **Position angle** (reference: XK axis)
- **Q**: **Number of corners**
- **K**: **Width across** – Inscribed circle diameter
- **Ki**: **Edge length**
- **BR**: **Cham. width** or **Rounding radius**

You can use the **Select reference plane** function to determine the **Reference dimension ZR**.

Further information: "Reference data and nested contours", Page 490

ICP generates the following:

- The **FRONT** section code with the **Reference dimension** parameter. In nested contours, **ICP** generates only one section code
- A **G308** with the parameters **Contour name** and **Milling depth**.
- A **G307** with the parameters of the figure
- A **G309**

Linear slot on front face

Reference data of face:

- **ID**: **Contour**
- **PT**: **Milling depth**
- **ZR**: **Reference dimension**

Parameters of figure:

- **XKM**, **YKM**: **Center point** of figure (in Cartesian coordinates)
- **MPX**: **Diameter** – **Center** (in polar coordinates)
- **MPC**: **Angle** – **Center** (in polar coordinates)
- **A**: **Position angle** (reference: XK axis)
- **K**: **Length**
- **B**: **Width**

You can use the **Select reference plane** function to determine the **Reference dimension ZR**.

Further information: "Reference data and nested contours", Page 490

ICP generates the following:

- The **FRONT** section code with the **Reference dimension** parameter. In nested contours, **ICP** generates only one section code
- A **G308** with the parameters **Contour name** and **Milling depth**.
- A **G301** with the parameters of the figure
- A **G309**
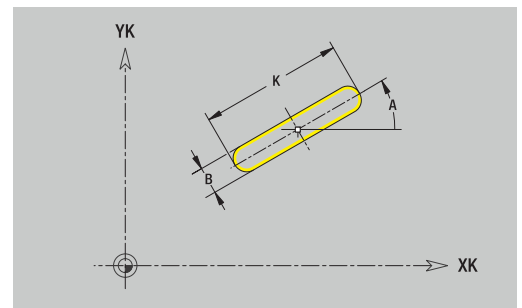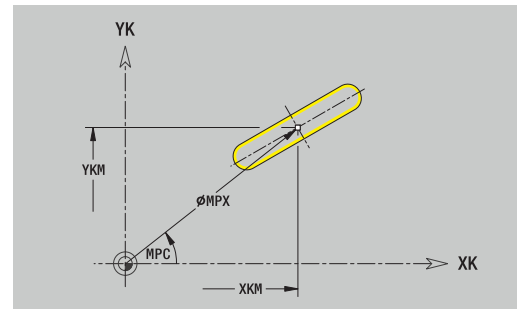
Circular slot on front face

Reference data of face:

- **ID**: **Contour**
- **PT**: **Milling depth**
- **ZR**: **Reference dimension**

Parameters of figure:

- **XKM**, **YKM**: **Center point** of figure (in Cartesian coordinates)
- **MPX**: **Diameter** – **Center** (in polar coordinates)
- **MPC**: **Angle** – **Center** (in polar coordinates)
- **A**: **Start angle** (reference: XK axis)
- **W**: **Final angle** (reference: XK axis)
- **R**: **Radius** – Curvature radius (reference: center point path of the slot)
- **Q2**: **Dir.rot.**
  - CW
  - CCW
- **B**: **Width**

You can use the **Select reference plane** function to determine the **Reference dimension ZR**.

Further information: "Reference data and nested contours", Page 490

ICP generates the following:

- The **FRONT** section code with the **Reference dimension** parameter. In nested contours, **ICP** generates only one section code
- A **G308** with the parameters **Contour name** and **Milling depth**.
- A **G302** or **G303** with the parameters of the figure
- A **G309**
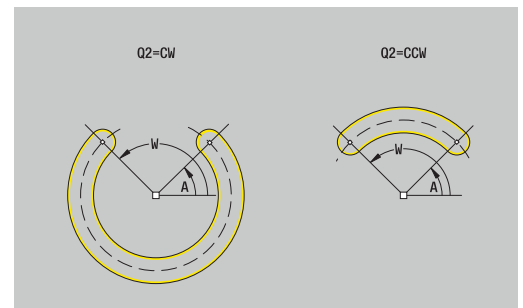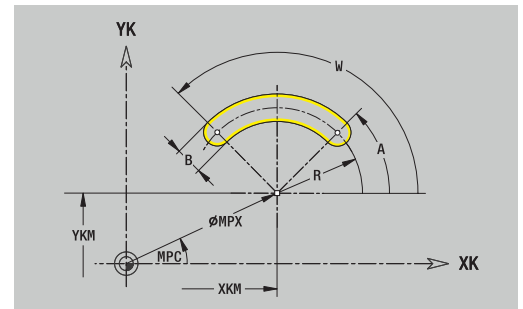
Hole on front face

This function defines a single hole that can contain the following elements:

- **Centering**
- **Hole**
- **Cavity**
- **Thread**

Reference data of hole:

- **ID**: **Contour**
- **ZR**: **Reference dimension**

Parameters of hole:

- **XKM**, **YKM**: **Center point** of hole (in Cartesian coordinates)
- **MPX**: **Diameter** – **Center** (in polar coordinates)
- **MPC**: **Angle** – **Center** (in polar coordinates)
- **Centering**
  - **O**: **Diameter**
- **Hole**
  - **B**: **Diameter**
  - **BT**: **Depth** (without algebraic sign)
  - **W**: **Angle**
- **Cavity**
  - **R**: **Diameter**
  - **U**: **Depth**
  - **E**: **Sink angle**
- **Thread**
  - **GD**: **Diameter**
  - **GT**: **Depth**
  - **K**: **Cut-in length**
  - **F**: **Thread pitch**
  - **GA**: **Thread direction**
    - **0: Right-hand thread**
    - **1: Left-hand thread**

You can use the **Select reference plane** function to determine the **Reference dimension ZR**.

Further information: "Reference data and nested contours", Page 490

ICP generates the following:

- The **FRONT** section code with the **Reference dimension** parameter. In nested contours, **ICP** generates only one section code
- An **G308** with the parameters **Contour name** and **Boring depth** (−1***BT**)
- A **G300** with the parameters of the hole
- A **G309**
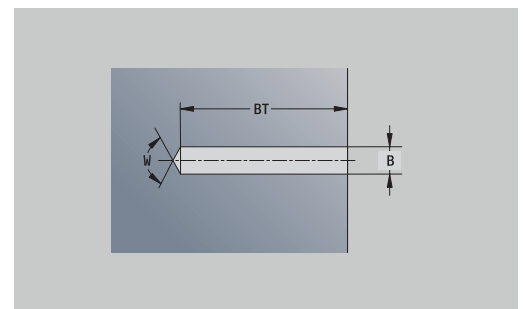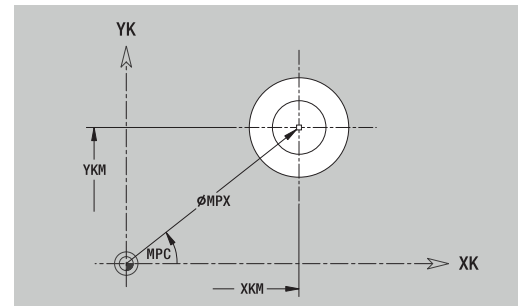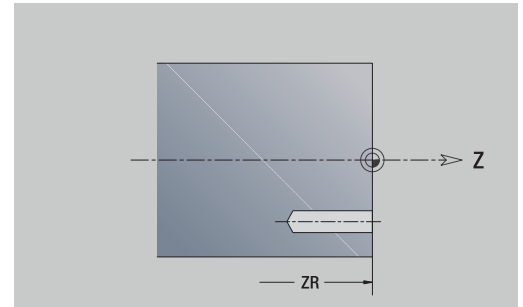
Linear pattern on front face

Reference data of face:

- **ID**: **Contour**
- **PT**: **Milling depth**
- **ZR**: **Reference dimension**

Parameters of pattern:

- **XK**, **YK**: **1st pattern point** (in Cartesian coordinates)
- **MX**: **Diameter** – **1st pattern point** (in polar coordinates)
- **MC**: **Angle** – **1st pattern point** (in polar coordinates)
- **QP**: **Quantity** of points in pattern
- **IP**, **JP**: **Final point** of pattern (in Cartesian coordinates)
- **IPi**, **JPi**: **Final point** – Distance between two pattern points (in **XK** and **YK**)
- **AP**: **Position angle**
- **RP**: **Length** – Total length of pattern
- **RPi**: **Length** – Distance between two pattern points
- Parameters of selected figure or hole

You can use the **Select reference plane** function to determine the **Reference dimension ZR**.

Further information: "Reference data and nested contours", Page 490

ICP generates the following:

- The **FRONT** section code with the **Reference dimension** parameter. In nested contours, **ICP** generates only one section code
- A **G308** with the parameters **Contour name** and **Milling depth** or **Boring depth** (−1***BT**)
- A **G401** with the parameters of the pattern
- The G code and the parameters of the figure or hole
- A **G309**
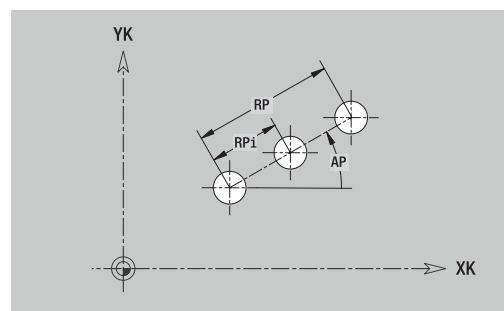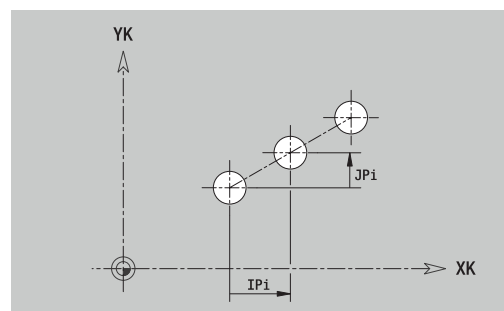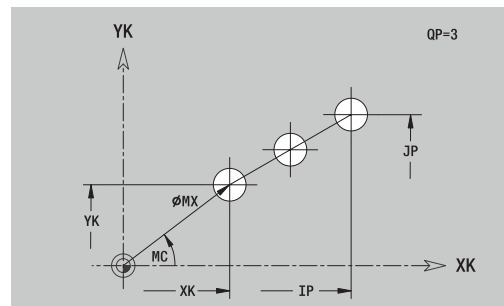
Circular pattern on front face

Reference data of face:

- **ID**: **Contour**
- **PT**: **Milling depth**
- **ZR**: **Reference dimension**

Parameters of pattern:

- **XK**, **YK**: **Center** of pattern (in Cartesian coordinates)
- **MX**: **Diameter** – **Center** of pattern (in polar coordinates)
- **MC**: **Angle** – **Center** of pattern (in polar coordinates)
- **QP**: **Quantity** of points in pattern
- **DR**: **Dir.rot.** (default: 0)
    - **DR** = 0, without **EP**: Figures are arranged on a full circle
    - **DR** = 0, with **EP**: Figures are arranged on the longer circular arc
    - **DR** = 0, with **EPi**: The algebraic sign of **EPi** defines the direction (**EPi** < 0: clockwise)
    - **DR** = 1, with **EP**: clockwise
    - **DR** = 1, with **EPi**: clockwise (algebraic sign of **EPi** has no effect)
    - **DR** = 2, with **EP**: counterclockwise
    - **DR** = 2, with **EPi**: counterclockwise (algebraic sign of **EPi** has no effect)
- **DP**: **Diameter**
- **AP**: **Start angle** (default: 0°)
- **EP**: **Final angle** (no input: the pattern elements are distributed evenly around 360°)
- **EPi**: **Final angle** – **Angle** between two figures
- **H**: **Element pos**
    - **0: Normal** – The figures are rotated about the circle center (rotation)
    - **1: Master** – The position of the figure relative to the coordinate system remains unchanged (translation)
- Parameters of the selected figure/hole

You can use the **Select reference plane** function to determine the **Reference dimension ZR**.

Further information: "Reference data and nested contours", Page 490

ICP generates the following:

- The **FRONT** section code with the **Reference dimension** parameter. In nested contours, **ICP** generates only one section code
- A **G308** with the parameters **Contour name** and **Milling depth** or **Boring depth** (−1*BT)
- A **G402** with the parameters of the pattern
- The **G** code and the parameters of the figure/hole
- A **G309**
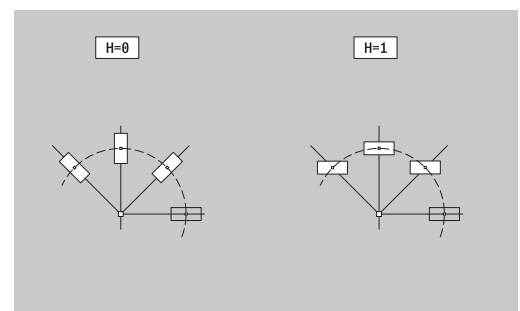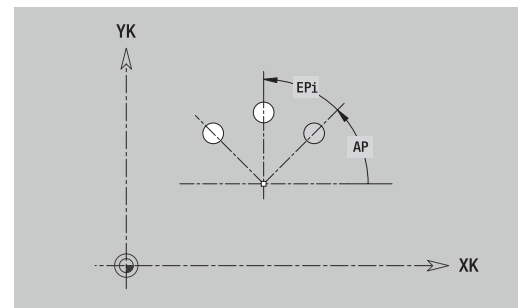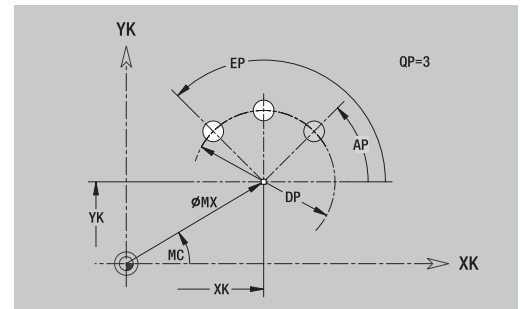
8.13 Lateral surface contours in smart.Turn operating mode (option 9)

In **smart.Turn** mode of operation, **ICP** provides the following contours for machining with the C axis:

- Complex contours defined with individual contour elements
- Figures
- Holes
- Pattern of figures or holes

Reference data of lateral surface

The reference data is followed by the contour definition with individual contour elements.

Further information: "Contour elements on lateral surface", Page 483

Reference data of lateral surface:

- **ID**: **Contour**
- **PT**: **Milling depth**
- **XR**: **Reference diameter**

You can use the **Select reference plane** function to determine the **Reference diameter XR**. The reference diameter is used to convert angle dimensions into linear dimensions.

"Reference data and nested contours"

ICP generates the following:

- The **LATERAL** section code with the **Reference diameter** parameter. In nested contours, **ICP** generates only one section code
- A **G308** with the parameters **Contour name** and **Milling depth**.
- A **G309** at the end of the contour description or after the figure

TURN PLUS attributes

In the TURN PLUS attributes, you can define settings for the **Automatic working plan generation** (**AWG**) submode.

Parameters for defining the starting point:

- **HC**: **Milling/drilling attribute**
    - **1: Contour milling G840/G847**
    - **2: Pocket milling G845/G848**
    - **3: Area milling G841-G844**
    - **4: Deburring G840**
    - **5: Engraving G801-G804**
    - **6: Contour + deburring G840/G847**
    - **7: Pocket + deburring G845/G848**
    - **8: face milling G797**
    - **9: face milling + deburring G797**
    - **10: bore milling G75**
    - **11: thread milling G799/G800/G806**
    - **12: bore and thread milling G75/G799..**
    - **14: Do not machine**
- **DF**: **Milling diameter**
- **QF**: **Cutter position**
    - **0: On the contour**
    - **1: Inside / left**
    - **2: Outside / right**
- **HF**: **Direction**
    - **0: Up-cut**
    - **1: Climb**
- **OF**: **Plunging behavior**
    - **0 / no input − vertical plunging**
    - **1: Helical plunging**
        - In roughing cycles, the tool plunges in a reciprocating movement when milling pockets, otherwise—i.e. when milling slots—in a helical movement.
        - In finishing cycles, the tool plunges in a 3-D approach arc when milling pockets.
    - **2: Reciprocating plunge**
        - In roughing cycles, the tool plunges in a reciprocating movement when milling pockets.
        - In finishing cycles, the tool plunges in a 3-D approach arc when milling pockets.
- **IF**: **Limit diameter**
- **RC**: **Trochoid width**
- **RB**: **Return plane**
- **WF**: **Angle** of chamfer
- **BF**: **Chamfer width**

Circle on lateral surface

Reference data of lateral surface:

- **ID**: **Contour**
- **PT**: **Milling depth**
- **XR**: **Reference diameter**

Parameters of figure:

- **ZM**: **Center**
- **CYM**: **Center** as linear dimension (reference: diameter **XR**)
- **CM**: **Center** (angle)
- **R**: **Radius**

You can use the **Select reference plane** function to determine the **Reference diameter XR**.

Further information: "Reference data and nested contours", Page 490

ICP generates the following:

- The **LATERAL** section code with the **Reference diameter** parameter. In nested contours, **ICP** generates only one section code
- A **G308** with the parameters **Contour name** and **Milling depth**.
- A **G314** with the parameters of the figure
- A **G309**

Rectangle on lateral surface

Reference data of lateral surface:

- **ID**: **Contour**
- **PT**: **Milling depth**
- **XR**: **Reference diameter**

Parameters of figure:

- **ZM**: **Center**
- **CYM**: **Center** as linear dimension (reference: diameter **XR**)
- **CM**: **Center** (angle)
- **A**: **Position angle**
- **K**: **Length**
- **B**: **Width**
- **BR**: **Cham. width** or **Rounding radius**

You can use the **Select reference plane** function to determine the **Reference diameter XR**.

Further information: "Reference data and nested contours", Page 490

ICP generates the following:

- The **LATERAL** section code with the **Reference diameter** parameter. In nested contours, **ICP** generates only one section code
- A **G308** with the parameters **Contour name** and **Milling depth**.
- A **G315** with the parameters of the figure
- A **G309**

Polygon on lateral surface

Reference data of lateral surface:

- **ID**: **Contour**
- **PT**: **Milling depth**
- **XR**: **Reference diameter**

Parameters of figure:

- **ZM**: **Center**
- **CYM**: **Center** as linear dimension (reference: diameter **XR**)
- **CM**: **Center** (angle)
- **A**: **Position angle**
- **Q**: **Number of corners**
- **K**: **Width across** – Inscribed circle diameter
- **Ki**: **Edge length**
- **BR**: **Cham. width** or **Rounding radius**

You can use the **Select reference plane** function to determine the **Reference diameter XR**.

Further information: "Reference data and nested contours", Page 490

ICP generates the following:

- The **LATERAL** section code with the **Reference diameter** parameter. In nested contours, **ICP** generates only one section code
- A **G308** with the parameters **Contour name** and **Milling depth**.
- A **G317** with the parameters of the figure
- A **G309**

Linear slot on lateral surface

Reference data of lateral surface:

- **ID**: **Contour**
- **PT**: **Milling depth**
- **XR**: **Reference diameter**

Parameters of figure:

- **ZM**: **Center**
- **CYM**: **Center** as linear dimension (reference: diameter **XR**)
- **CM**: **Center** (angle)
- **A**: **Position angle**
- **K**: **Length**
- **B**: **Width**

You can use the **Select reference plane** function to determine the **Reference diameter XR**.

Further information: "Reference data and nested contours", Page 490

ICP generates the following:

- The **LATERAL** section code with the **Reference diameter** parameter. In nested contours, **ICP** generates only one section code
- A **G308** with the parameters **Contour name** and **Milling depth**.
- A **G311** with the parameters of the figure
- A **G309**
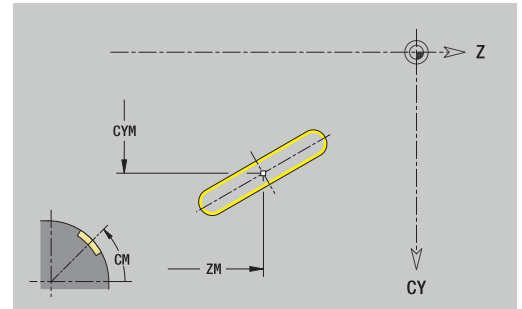
Circular slot on lateral surface

Reference data of lateral surface:

- **ID**: **Contour**
- **PT**: **Milling depth**
- **XR**: **Reference diameter**

Parameters of figure:

- **ZM**: **Center**
- **CYM**: **Center** as linear dimension (reference: diameter **XR**)
- **CM**: **Center** (angle)
- **A**: **Start angle**
- **W**: **Final angle**
- **R**: **Radius** – Curvature radius (reference: center point path of the slot)
- **Q2**: **Dir.rot.**
  - CW
  - CCW
- **B**: **Width**

You can use the **Select reference plane** function to determine the **Reference diameter XR**.

Further information: "Reference data and nested contours", Page 490

ICP generates the following:

- The **LATERAL** section code with the **Reference diameter** parameter. In nested contours, **ICP** generates only one section code
- A **G308** with the parameters **Contour name** and **Milling depth**.
- A **G312** or **G313** with the parameters of the figure
- A **G309**
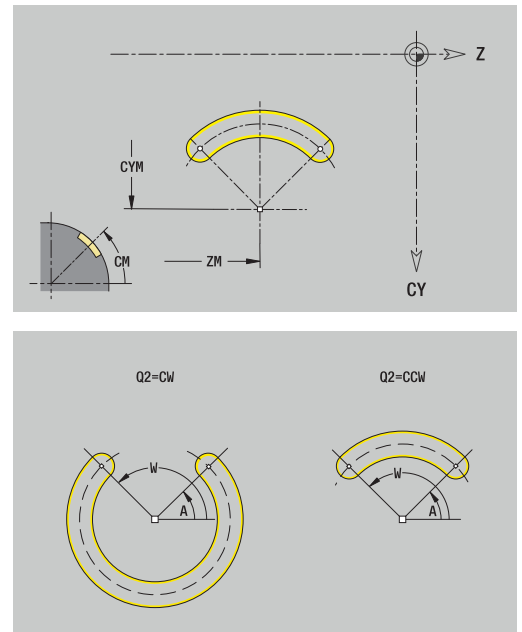
Hole on lateral surface

This function defines a single hole that can contain the following elements:

- **Centering**
- **Hole**
- **Cavity**
- **Thread**

Reference data of hole:

- **ID**: **Contour**
- **X**: **Reference dimension**

Parameters of hole:

- **ZM**: **Center**
- **CYM**: **Center** as linear dimension (reference: diameter **XR**)
- **CM**: **Center** (angle)
- **Centering**
  - **O**: **Diameter**
- **Hole**
  - **B**: **Diameter**
  - **BT**: **Depth** (without algebraic sign)
  - **W**: **Angle**
- **Cavity**
  - **R**: **Diameter**
  - **U**: **Depth**
  - **E**: **Sink angle**
- **Thread**
  - **GD**: **Diameter**
  - **GT**: **Depth**
  - **K**: **Cut-in length**
  - **F**: **Thread pitch**
  - **GA**: **Thread direction**
    - **0: Right-hand thread**
    - **1: Left-hand thread**

You can use the **Select reference plane** function to determine the **Reference dimension XR**.

Further information: "Reference data and nested contours", Page 490

ICP generates the following:

- The **LATERAL** section code with the **Reference dimension** parameter. In nested contours, **ICP** generates only one section code
- A **G308** with the parameters **Contour name** and **Boring depth** (−1*BT)
- A **G310** with the parameters of the hole
- A **G309**
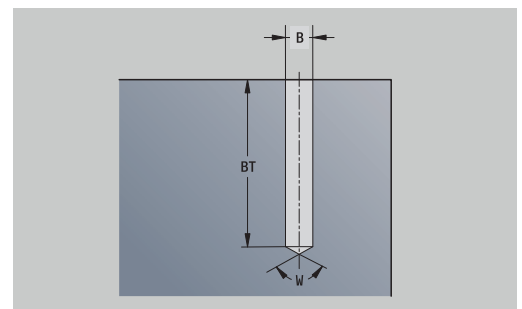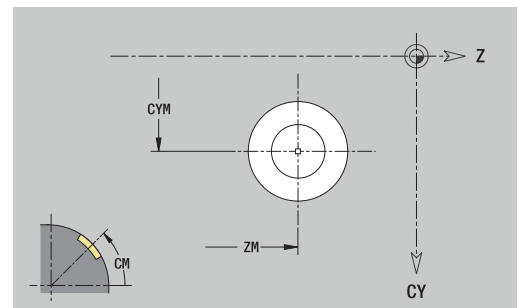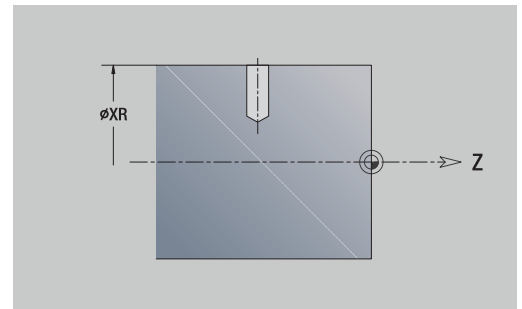
Linear pattern on lateral surface

Reference data of lateral surface:

- **ID**: **Contour**
- **PT**: **Milling depth**
- **XR**: **Reference diameter**

Parameters of figure:

- **Z**: **1st pattern point**
- **CY**: **1st pattern point** as linear dimension (reference: diameter **XR**)
- **C**: **1st pattern point** (angle)
- **QP**: **Quantity** of points in pattern
- **ZE**: **Final point** of pattern
- **ZEi**: **Final point** – Distance between two pattern points (in **Z**)
- **WP**: **Final point** of pattern (angle)
- **WPi**: **Final point** – Distance between two pattern points (angle)
- **AP**: **Position angle**
- **RP**: **Length** – Total length of pattern
- **RPi**: **Length** – Distance between two pattern points
- Parameters of the selected figure/hole

You can use the **Select reference plane** function to determine the **Reference diameter XR**.

Further information: "Reference data and nested contours", Page 490

ICP generates the following:

- The **LATERAL** section code with the **Reference diameter** parameter. In nested contours, **ICP** generates only one section code
- A **G308** with the parameters **Contour name** and **Milling depth** or **Boring depth** (−1***BT**)
- A **G411** with the parameters of the pattern
- The **G** code and the parameters of the figure or hole
- A **G309**
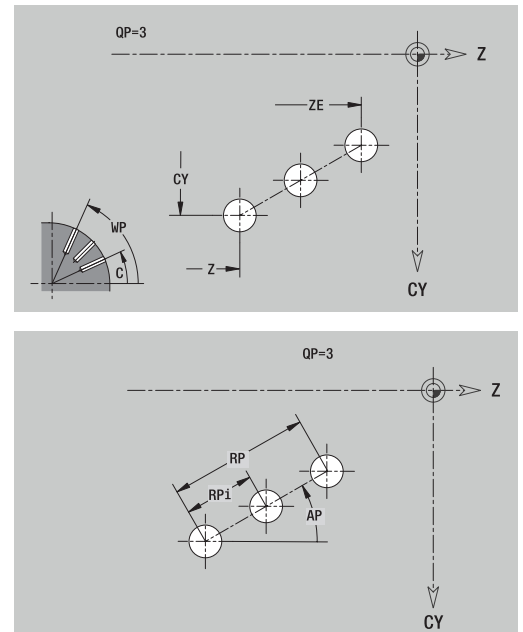
Circular pattern on lateral surface

Reference data of lateral surface:

- **ID**: **Contour**
- **PT**: **Milling depth**
- **XR**: **Reference diameter**

Parameters of figure:

- **Z**: **Center** of pattern
- **CY**: **Center** of pattern as linear dimension (reference: diameter **XR**)
- **C**: **Center** of pattern (angle)
- **QP**: **Quantity** of points in pattern
- **DR**: **Dir.rot.** (default: 0)
    - **DR** = 0, without **EP**: Figures are arranged on a full circle
    - **DR** = 0, with **EP**: Figures are arranged on the longer circular arc
    - **DR** = 0, with **EPi**: The algebraic sign of **EPi** defines the direction (**EPi** < 0: clockwise)
    - **DR** = 1, with **EP**: clockwise
    - **DR** = 1, with **EPi**: clockwise (algebraic sign of **EPi** has no effect)
    - **DR** = 2, with **EP**: counterclockwise
    - **DR** = 2, with **EPi**: counterclockwise (algebraic sign of **EPi** has no effect)
- **DP**: **Diameter**
- **AP**: **Start angle** (default: 0°)
- **EP**: **Final angle** (no input: the pattern elements are distributed evenly around 360°)
- **EPi**: **Final angle** – **Angle** between two figures
- **H**: **Element pos**
    - **0: Normal** – The figures are rotated about the circle center (rotation)
    - **1: Master** – The position of the figure relative to the coordinate system remains unchanged (translation)

You can use the **Select reference plane** function to determine the **Reference diameter XR**.

Further information: "Reference data and nested contours", Page 490

ICP generates the following:

- The **LATERAL** section code with the **Reference diameter** parameter. In nested contours, **ICP** generates only one section code
- A **G308** with the parameters **Contour name** and **Milling depth** or **Boring depth** (−1***BT**)
- A **G412** with the parameters of the pattern
- The **G** code and the parameters of the figure/hole
- A **G309**
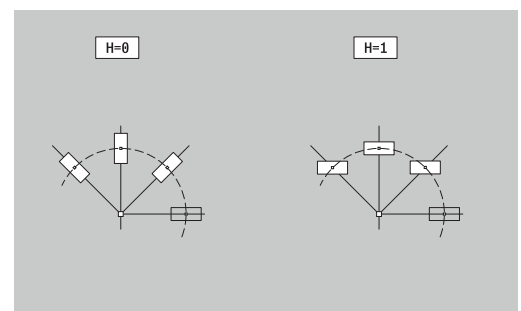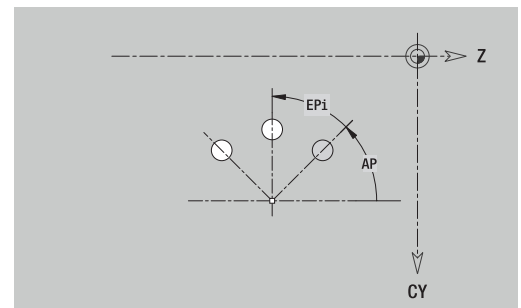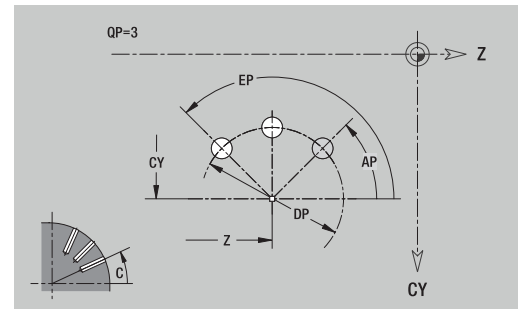
8.14 Contours in the XY plane

In **smart.Turn** mode of operation, **ICP** provides the following contours for machining with the Y axis:

- Complex contours defined with individual contour elements
- Figures
- Holes
- Pattern of figures or holes
- Single surface
- Polygon

Enter the dimensions of the XY plane contour elements in Cartesian or polar coordinates. You can switch between them by soft key. You can mix Cartesian coordinates and polar coordinates to define a point.

Soft keys for polar coordinates

	Switch the field to entering the angle **W**
	Switch the field to entering the radius **P**

Reference data in XY plane

The reference data is followed by the contour definition with individual contour elements.

Reference data of milling operations:

- **ID**: **Contour**
- **PT**: **Milling depth**
- **C**: **Spindle angle**
- **IR**: **Limit diameter**
- **ZR**: **Reference dimension**

You can use the **Select reference plane** function to determine the **Reference dimension ZR** and the **Limit diameter IR**.

Further information: "Reference data and nested contours", Page 490

ICP generates the following:

- The **FRONT Y** section code with the **Reference dimension**, **Spindle angle**, and **Limit diameter** parameters. The section code is omitted for nested contours.
- A **G308** with the parameters **Contour name** and **Milling depth**.
- A **G309** at the end of the contour description

TURN PLUS attributes

In the TURN PLUS attributes you can define settings for the **automatic working plan generation** (**AWG**) submode.

Parameters for defining the starting point:

- **HC**: **Milling/drilling attribute**
  - **1: Contour milling G840/G847**
  - **2: Pocket milling G845/G848**
  - **3: Area milling G841-G844**
  - **4: Deburring G840**
  - **5: Engraving G801-G804**
  - **6: Contour + deburring G840/G847**
  - **7: Pocket + deburring G845/G848**
  - **8: face milling G797**
  - **9: face milling + deburring G797**
  - **10: bore milling G75**
  - **11: thread milling G799/G800/G806**
  - **12: bore and thread milling G75/G799..**
  - **14: Do not machine**
- **DF**: **Milling diameter**
- **QF**: **Cutter position**
  - **0: On the contour**
  - **1: Inside / left**
  - **2: Outside / right**
- **HF**: **Direction**
  - **0: Up-cut**
  - **1: Climb**
- **OF**: **Plunging behavior**
  - **0 / no input – vertical plunging**
  - **1: Helical plunging**
    - In roughing cycles, the tool plunges in a reciprocating movement when milling pockets, otherwise—i.e. when milling slots—in a helical movement.
    - In finishing cycles, the tool plunges in a 3-D approach arc when milling pockets.
  - **2: Reciprocating plunge**
    - In roughing cycles, the tool plunges in a reciprocating movement when milling pockets.
    - In finishing cycles, the tool plunges in a 3-D approach arc when milling pockets.
- **IF**: **Limit diameter**
- **RC**: **Trochoid width**
- **RB**: **Return plane**
- **WF**: **Angle** of chamfer
- **BF**: **Chamfer width**

Basic elements of XY plane

Starting point of contour in XY plane

Enter the coordinates for the Starting point and Destinat. point in the first contour element of the contour. Entering the starting point is only possible in the first contour element. In subsequent contour elements, the Starting point results from the previous contour element in each case.

To define the Starting point:

▶ Select the **Contour** menu item

Insert
element

▶ Alternative: Press the **Insert element** soft key

▶ Define the Starting point

Parameters for defining the starting point:

- **XS**, **YS**: **Starting point** of contour
- **WS**: **Starting point** of contour (angle in polar coordinates)
- **PS**: **Starting point** of contour (radius in polar coordinates)

ICP generates a **G170** in **smart.Turn** mode of operation.

Vertical lines in XY plane

To program a vertical line:

▶ Select the line direction

▶ Enter the line dimensions

▶ Define the transition to the next contour element

Parameters:

- **Y**: **Target point**
- **Yi**: **Target point** (incremental)
- **W**: **Target point** (angle in polar coordinates)
- **P**: **Target point** (in polar coordinates)
- **L**: **Line length**
- **F**: **Further information:** "Machining attributes", Page 438

ICP generates a **G171** in **smart.Turn** mode of operation.
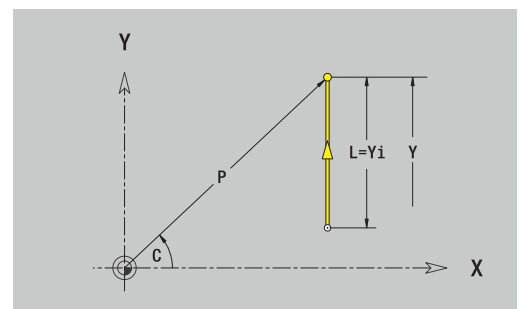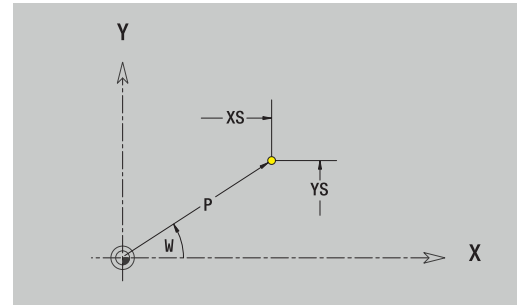
Horizontal lines in XY plane

To program a horizontal line:

▶ Select the line direction

▶ Enter the line dimensions
▶ Define the transition to the next contour element

Parameters:
- **X**: **Target point**
- **Xi**: **Target point** (incremental)
- **W**: **Target point** (angle in polar coordinates)
- **P**: **Target point** (in polar coordinates)
- **L**: **Line length**
- **F**: **Further information:** "Machining attributes", Page 438

ICP generates a **G171** in **smart.Turn** mode of operation.
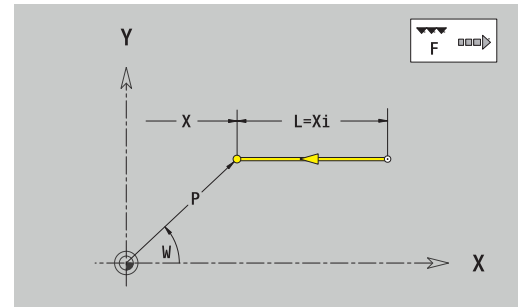
Line at angle in XY plane

To program a line at an angle:

▶ Select the line direction

▶ Enter the line dimensions
▶ Define the transition to the next contour element

Parameters:
- **X**, **Y**: **Target point**
- **Xi**, **Yi**: **Target point** (incremental)
- **W**: **Target point** (angle in polar coordinates)
- **P**: **Target point** (in polar coordinates)
- **L**: **Line length**
- **AN**: **Angle**
- **ANn**: **Angle** to the subsequent element
- **ANp**: **Angle** to the previous element
- **F**: **Further information:** "Machining attributes", Page 438

ICP generates a **G171** in **smart.Turn** mode of operation.
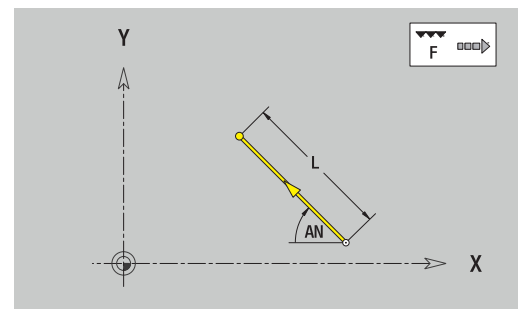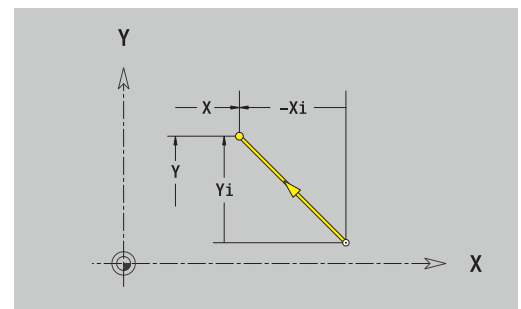
Circular arc in XY plane

To program a circular arc:

▶ Select the arc's direction of rotation

▶ Enter the arc dimensions

▶ Define the transition to the next contour element

Parameters:

- **X**, **Y**: **Target point**
- **Xi**, **Yi**: **Target point** (incremental)
- **P**: **Target point** (in polar coordinates)
- **Pi**: **Target point** – Distance between starting point and target point (polar, incremental)
- **W**: **Target point** (angle in polar coordinates)
- **Wi**: **Target point** (polar angle, incremental; reference: starting point)
- **I**, **J**: **Center point** of circular arc
- **Ii**, **Ji**: **Center point** of circular arc (incremental) – Distance between starting point and **Center point** in **X** and **Y**
- **PM**: **Center point** of circular arc (in polar coordinates)
- **PMi**: **Center point** of circular arc – Distance between starting point and **Center point** (polar, incremental)
- **WM**: **Center point** of circular arc (polar angle)
- **WMi**: **Center point** of circular arc (polar angle, incremental; reference: starting point)
- **R**: **Radius**
- **ANs**: **Angle** – Tangential angle in the starting point
- **ANe**: **Angle** – Tangential angle at the target point
- **ANn**: **Angle** to the subsequent element
- **ANp**: **Angle** to the previous element
- **F**: **Further information:** "Machining attributes", Page 438

ICP generates a **G172** or **G173** in **smart.Turn** mode of operation.
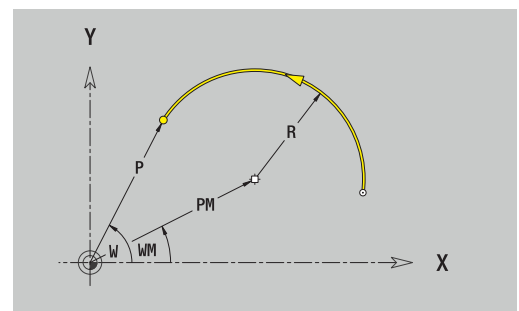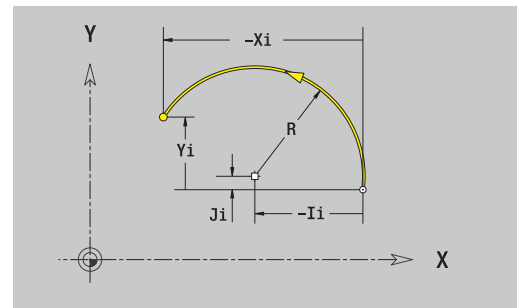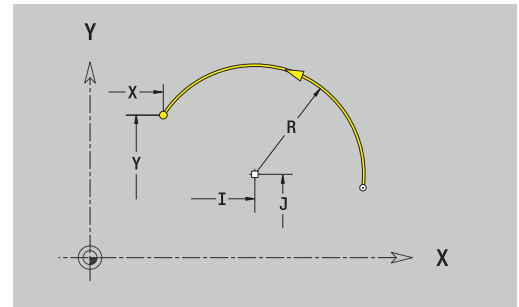
Form elements of XY plane

Chamfer or rounding in XY plane

To program a chamfer or rounding radius:

▶ Select the form element menu

▶ Select the Chamfer menu item

▶ Enter the **Chamfer width BR**

▶ Alternative: Select a rounding radius

▶ Enter the **Rounding radius BR**

▶ Enter the chamfer or rounding as first contour element: **Position AN**

Parameters:

- **BR**: **Cham. width** or **Rounding radius**
- **AN**: **Element pos**
- **F**: **Further information:** "Machining attributes", Page 438

Chamfers or roundings are defined at contour corners. A **contour corner** is the intersection between the approaching contour element and the departing contour element. A chamfer or rounding cannot be calculated until the basic contour element is known.

ICP integrates the chamfer or rounding in the **G171**, **G172** or **G173** basic element in **smart.Turn** mode of operation.

Contour begins with a chamfer or a rounding: Enter the position of the **intended corner** as starting point. Then, in the form element menu, select Chamfer or Rounding. Since the **approaching contour element** is missing, enter the **Position AN** to clearly define the position of the chamfer or rounding.

ICP converts a chamfer or rounding at the start of the contour to a linear or circular element.
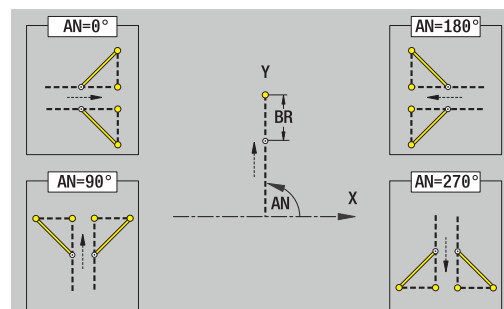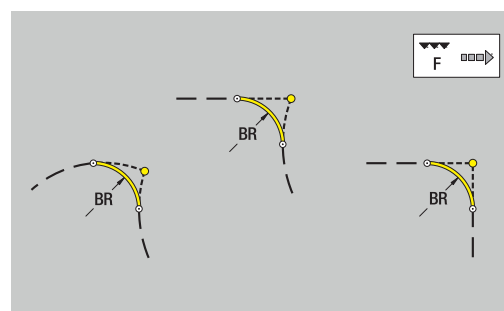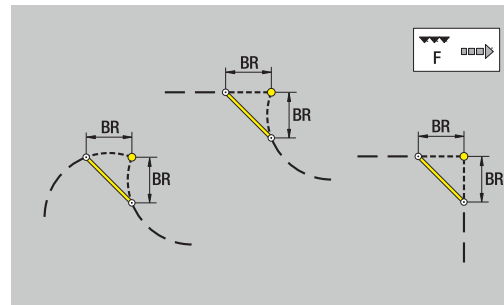
Figures, patterns and holes in XY plane (front face)

Circle in XY plane

Reference data of XY plane:

- **ID**: **Contour**
- **PT**: **Milling depth**
- **C**: **Spindle angle**
- **IR**: **Limit diameter**
- **ZR**: **Reference dimension**

Parameters of figure:

- **XM**, **YM**: **Center**
- **R**: **Radius**

You can use the **Select reference plane** function to determine the **Reference dimension ZR** and the **Limit diameter IR**.

Further information: "Reference data and nested contours", Page 490

ICP generates the following:

- The **FRONT Y** section code with the **Limit diameter**, **Reference dimension**, and **Spindle angle** parameters. The section code is omitted for nested contours.
- A **G308** with the parameters **Contour name** and **Milling depth**.
- A **G374** with the parameters of the figure
- A **G309**

Rectangle in XY plane

Reference data of XY plane:

- **ID**: **Contour**
- **PT**: **Milling depth**
- **C**: **Spindle angle**
- **IR**: **Limit diameter**
- **ZR**: **Reference dimension**

Parameters of figure:

- **XM**, **YM**: **Center**
- **A**: **Position angle**
- **K**: **Length**
- **B**: **Width**
- **BR**: **Cham. width** or **Rounding radius**

You can use the **Select reference plane** function to determine the **Reference dimension ZR** and the **Limit diameter IR**.

Further information: "Reference data and nested contours", Page 490

ICP generates the following:

- The **FRONT Y** section code with the **Limit diameter**, **Reference dimension**, and **Spindle angle** parameters. The section code is omitted for nested contours.
- A **G308** with the parameters **Contour name** and **Milling depth**.
- A **G375** with the parameters of the figure
- A **G309**

Polygon in XY plane

Reference data of XY plane:

- **ID**: **Contour**
- **PT**: **Milling depth**
- **C**: **Spindle angle**
- **IR**: **Limit diameter**
- **ZR**: **Reference dimension**

Parameters of figure:

- **XM**, **YM**: **Center**
- **A**: **Position angle**
- **Q**: **Number of corners**
- **K**: **Width across** – Inscribed circle diameter
- **Ki**: **Edge length**
- **BR**: **Cham. width** or **Rounding radius**

You can use the **Select reference plane** function to determine the **Reference dimension ZR** and the **Limit diameter IR**.

Further information: "Reference data and nested contours", Page 490

ICP generates the following:

- The **FRONT Y** section code with the **Limit diameter**, **Reference dimension**, and **Spindle angle** parameters. The section code is omitted for nested contours.
- A **G308** with the parameters **Contour name** and **Milling depth**.
- A **G377** with the parameters of the figure
- A **G309**

Linear slot in XY plane

Reference data of XY plane:

- **ID**: **Contour**
- **PT**: **Milling depth**
- **C**: **Spindle angle**
- **IR**: **Limit diameter**
- **ZR**: **Reference dimension**

Parameters of figure:

- **XM**, **YM**: **Center**
- **A**: **Position angle**
- **K**: **Length**
- **B**: **Width**

You can use the **Select reference plane** function to determine the **Reference dimension ZR** and the **Limit diameter IR**.

Further information: "Reference data and nested contours", Page 490

ICP generates the following:

- The **FRONT Y** section code with the **Limit diameter**, **Reference dimension**, and **Spindle angle** parameters. The section code is omitted for nested contours.
- A **G308** with the parameters **Contour name** and **Milling depth**.
- A **G371** with the parameters of the figure
- A **G309**
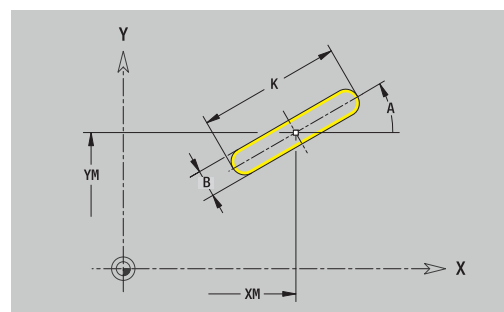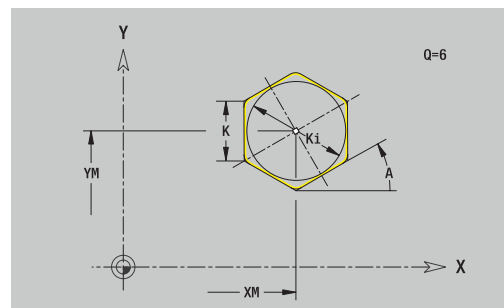
Circular slot in XY plane

Reference data of XY plane:

- **ID**: **Contour**
- **PT**: **Milling depth**
- **C**: **Spindle angle**
- **IR**: **Limit diameter**
- **ZR**: **Reference dimension**

Parameters of figure:

- **XM**, **YM**: **Center**
- **A**: **Start angle**
- **W**: **Final angle**
- **R**: **Radius** – Curvature radius (reference: center point path of the slot)
- **Q2**: **Dir.rot.**
  - **CW**
  - **CCW**
- **B**: **Width**

You can use the **Select reference plane** function to determine the **Reference dimension ZR** and the **Limit diameter IR**.

Further information: "Reference data and nested contours", Page 490

ICP generates the following:

- The **FRONT Y** section code with the **Limit diameter**, **Reference dimension**, and **Spindle angle** parameters. The section code is omitted for nested contours.
- A **G308** with the parameters **Contour name** and **Milling depth**.
- A **G372** or **G373** with the parameters of the figure
- A **G309**
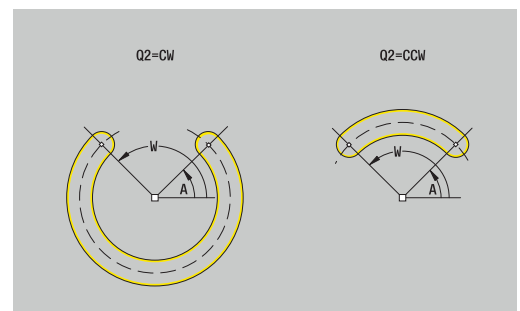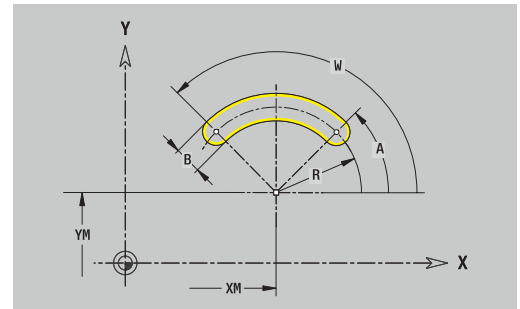
Hole in XY plane

This function defines a single hole that can contain the following elements:

- **Centering**
- **Hole**
- **Cavity**
- **Thread**

Reference data of hole:

- **ID**: **Contour**
- **C**: **Spindle angle**
- **IR**: **Limit diameter**
- **ZR**: **Reference dimension**

Parameters of hole:

- **XM**, **YM**: **Center**
- **Centering**
    - **O**: **Diameter**
- **Hole**
    - **B**: **Diameter**
    - **BT**: **Depth** (without algebraic sign)
    - **W**: **Angle**
- **Cavity**
    - **R**: **Diameter**
    - **U**: **Depth**
    - **E**: **Sink angle**
- **Thread**
    - **GD**: **Diameter**
    - **GT**: **Depth**
    - **K**: **Cut-in length**
    - **F**: **Thread pitch**
    - **GA**: **Thread direction**
        - **0: Right-hand thread**
        - **1: Left-hand thread**

You can use the **Select reference plane** function to determine the **Reference dimension ZR** and the **Limit diameter IR**.

Further information: "Reference data and nested contours", Page 490

ICP generates the following:

- The **FRONT Y** section code with the **Limit diameter**, **Reference dimension**, and **Spindle angle** parameters. The section code is omitted for nested contours.
- A **G308** with the parameters **Contour name** and **Boring depth** (−1*BT)
- A **G370** with the parameters of the hole
- A **G309**
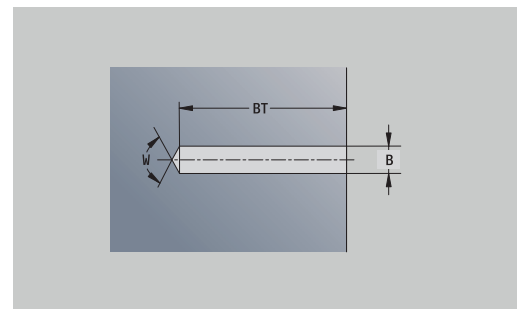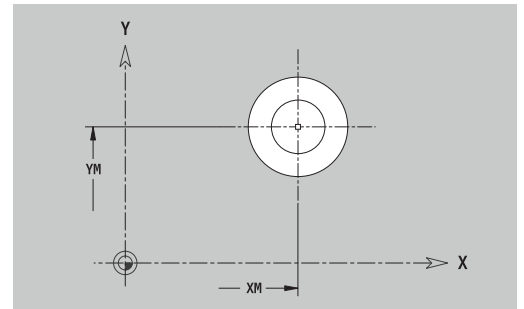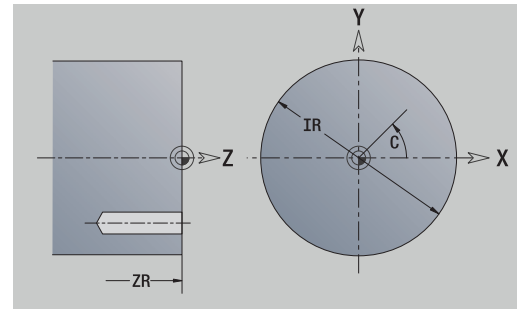
Linear pattern in XY plane

Reference data of XY plane:

- **ID**: **Contour**
- **PT**: **Milling depth**
- **C**: **Spindle angle**
- **IR**: **Limit diameter**
- **ZR**: **Reference dimension**

Parameters of pattern:

- **X**, **Y**: **1st pattern point**
- **QP**: **Quantity** of points in pattern
- **IP**, **JP**: **Final point** of pattern (in Cartesian coordinates)
- **IPi**, **JPi**: **Final point** – Distance between two pattern points (in **X** and **Y**)
- **AP**: **Position angle**
- **RP**: **Length** – Total length of pattern
- **RPi**: **Length** – Distance between two pattern points
- Parameters of the selected figure/hole

You can use the **Select reference plane** function to determine the **Reference dimension ZR** and the **Limit diameter IR**.

Further information: "Reference data and nested contours", Page 490

ICP generates the following:

- The **FRONT Y** section code with the **Limit diameter**, **Reference dimension**, and **Spindle angle** parameters. The section code is omitted for nested contours.
- A **G308** with the parameters **Contour name** and **Milling depth** or **Boring depth** (−1*BT)
- A **G471** with the parameters of the pattern
- The G code and the parameters of the figure/hole
- A **G309**
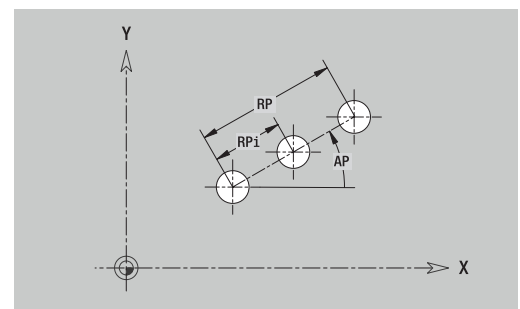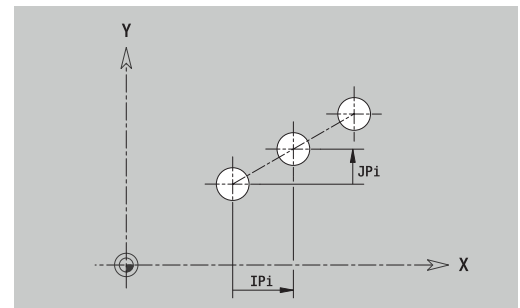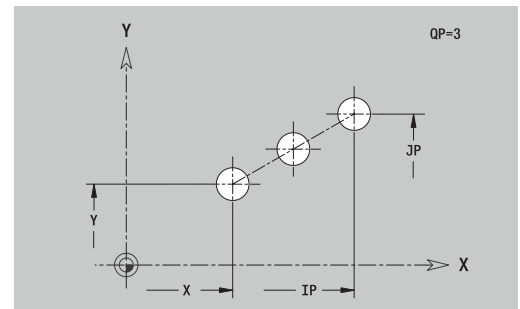
Circular pattern in XY plane

Reference data of XY plane:

- **ID**: **Contour**
- **PT**: **Milling depth**
- **C**: **Spindle angle**
- **IR**: **Limit diameter**
- **ZR**: **Reference dimension**

Parameters of pattern:

- **X**, **Y**: **Center** of pattern
- **QP**: **Quantity** of points in pattern
- **DR**: **Dir.rot.** (default: 0)
    - **DR** = 0, without **EP**: Figures are arranged on a full circle
    - **DR** = 0, with **EP**: Figures are arranged on the longer circular arc
    - **DR** = 0, with **EPi**: The algebraic sign of **EPi** defines the direction (**EPi** < 0: clockwise)
    - **DR** = 1, with **EP**: clockwise
    - **DR** = 1, with **EPi**: clockwise (algebraic sign of **EPi** has no effect)
    - **DR** = 2, with **EP**: counterclockwise
    - **DR** = 2, with **EPi**: counterclockwise (algebraic sign of **EPi** has no effect)
- **DP**: **Diameter**
- **AP**: **Start angle** (default: 0°)
- **EP**: **Final angle** (no input: the pattern elements are distributed evenly around 360°)
- **EPi**: **Final angle** − **Angle** between two figures
- **H**: **Element pos**
    - **0: Normal** − The figures are rotated about the circle center (rotation)
    - **1: Master** − The position of the figure relative to the coordinate system remains unchanged (translation)
- Parameters of the selected figure/hole

You can use the **Select reference plane** function to determine the **Reference dimension ZR** and the **Limit diameter IR**.

Further information: "Reference data and nested contours", Page 490

ICP generates the following:

- The **FRONT Y** section code with the **Limit diameter**, **Reference dimension**, and **Spindle angle** parameters. The section code is omitted for nested contours.
- A **G308** with the parameters **Contour name** and **Milling depth** or **Boring depth** (−1*BT)
- A **G472** with the parameters of the pattern
- The G code and the parameters of the figure/hole
- A **G309**
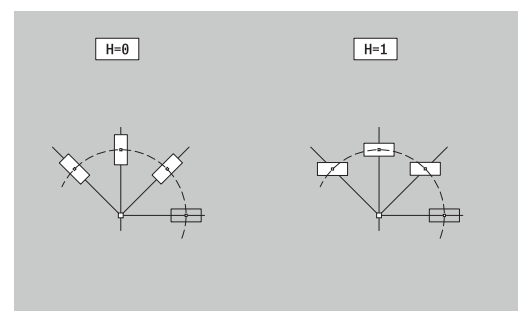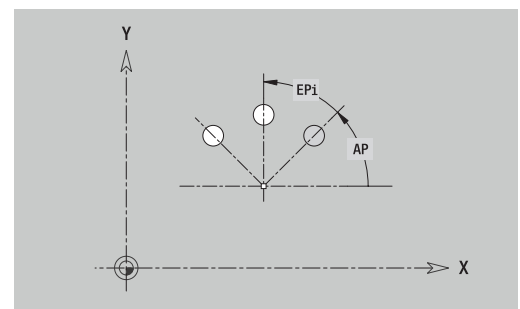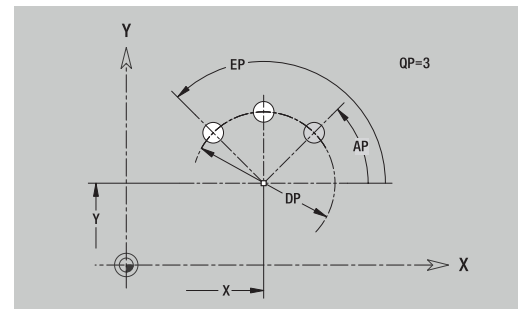
Single surface in XY plane

This function defines a surface in the XY plane.

Reference data of XY plane:

- **ID**: **Contour**
- **C**: **Spindle angle**
- **IR**: **Limit diameter**

Parameters of single surface:

- **Z**: **Reference edge**
- **Ki**: **Depth**
- **K**: **Residual depth**
- **B**: **Width** (reference: **Reference dimension ZR**)
    - **B** < 0: Surface in negative Z direction
    - **B** > 0: Surface in positive Z direction

You can switch between **Depth Ki** and **Residual depth K** by soft key.

You can use the **Select reference plane** function to determine the **Reference dimension ZR** and the **Limit diameter IR**.

Further information: "Reference data and nested contours", Page 490

ICP generates the following:

- The **FRONT Y** section code with the **Limit diameter**, **Reference dimension**, and **Spindle angle** parameters. The section code is omitted for nested contours.
- A **G308** with the parameter **Contour name**.
- A **G376** with the parameters of the single surface
- A **G309**

Soft key

Residual depth	Switch the field to the input of **Residual depth K**
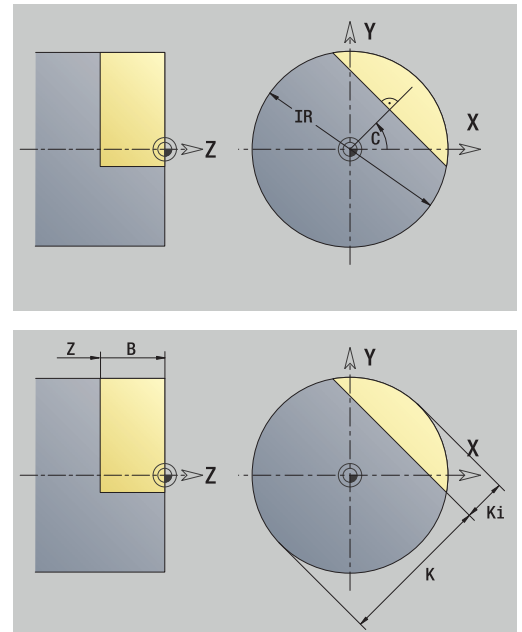
Centric polygon in XY plane

This function defines polygonal surfaces in the XY plane.

Reference data of XY plane:

- **ID**: **Contour**
- **C**: **Spindle angle**
- **IR**: **Limit diameter**

Parameters of single surface:

- **Z**: **Reference edge**
- **Q**: **No. of surfaces** (**Q** >= 2)
- **K**: **Width across** – Inscribed circle diameter
- **Ki**: **Edge length**
- **B**: **Width** (reference: **Reference dimension ZR**)
    - **B** < 0: Surface in negative Z direction
    - **B** > 0: Surface in positive Z direction

You can switch between **Edge length Ki** and **Width across K** by soft key.

You can use the **Select reference plane** function to determine the **Reference dimension ZR** and the **Limit diameter IR**.

Further information: "Reference data and nested contours", Page 490

ICP generates the following:

- The **FRONT Y** section code with the **Limit diameter**, **Reference dimension**, and **Spindle angle** parameters. The section code is omitted for nested contours.
- A **G308** with the parameter **Contour name**.
- A **G477** with the parameters of the polygon
- A **G309**

Soft key
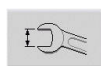
🔧	Switch the field to the input of **Width across K**
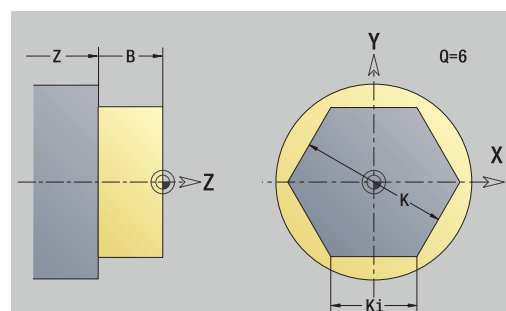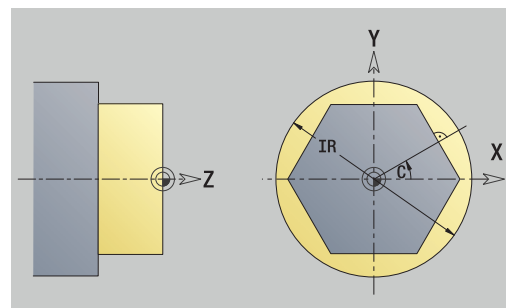
8.15 Contours in the YZ plane

In **smart.Turn** mode of operation, **ICP** provides the following contours for machining with the Y axis:

- Complex contours defined with individual contour elements
- Figures
- Holes
- Pattern of figures or holes
- Single surface
- Polygon

Enter the dimensions of the YZ plane contour elements in Cartesian or polar coordinates. You can switch between them by soft key. You can mix Cartesian coordinates and polar coordinates to define a point.

Soft keys for polar coordinates

	Switch the field to entering the angle **W**
	Switch the field to entering the radius **P**

Reference data in YZ plane

The reference data is followed by the contour definition with individual contour elements.

Reference data of milling operations:

- **ID**: **Contour**
- **PT**: **Milling depth**
- **C**: **Spindle angle**
- **XR**: **Reference diameter**

You can use the **Select reference plane** function to determine the **Reference diameter XR**.

Further information: "Reference data and nested contours", Page 490

ICP generates the following:

- The **SURFACE Y** section code with the **Reference diameter** and **Spindle angle** parameters. The section code is omitted for nested contours.
- A **G308** with the parameters **Contour name** and **Milling depth**
- A **G309** at the end of the contour description

TURN PLUS attributes

In the TURN PLUS attributes, you can define settings for the **Automatic working plan generation** (**AWG**) submode.

Parameters for defining the starting point:

- **HC**: **Milling/drilling attribute**
  - **1: Contour milling G840/G847**
  - **2: Pocket milling G845/G848**
  - **3: Area milling G841-G844**
  - **4: Deburring G840**
  - **5: Engraving G801-G804**
  - **6: Contour + deburring G840/G847**
  - **7: Pocket + deburring G845/G848**
  - **8: face milling G797**
  - **9: face milling + deburring G797**
  - **10: bore milling G75**
  - **11: thread milling G799/G800/G806**
  - **12: bore and thread milling G75/G799..**
  - **14: Do not machine**
- **DF**: **Milling diameter**
- **QF**: **Cutter position**
  - **0: On the contour**
  - **1: Inside / left**
  - **2: Outside / right**
- **HF**: **Direction**
  - **0: Up-cut**
  - **1: Climb**
- **OF**: **Plunging behavior**
  - **0 / no input – vertical plunging**
  - **1: Helical plunging**
    - In roughing cycles, the tool plunges in a reciprocating movement when milling pockets, otherwise—i.e. when milling slots—in a helical movement.
    - In finishing cycles, the tool plunges in a 3-D approach arc when milling pockets.
  - **2: Reciprocating plunge**
    - In roughing cycles, the tool plunges in a reciprocating movement when milling pockets.
    - In finishing cycles, the tool plunges in a 3-D approach arc when milling pockets.
- **IF**: **Limit diameter**
- **RC**: **Trochoid width**
- **RB**: **Return plane**
- **WF**: **Angle** of chamfer
- **BF**: **Chamfer width**

Basic elements of YZ plane

Starting point of contour in YZ plane

Enter the coordinates for the starting point and target point in the first contour element. Entering the starting point is only possible in the first contour element. In subsequent contour elements, the starting point results from the previous contour element in each case.

To define the starting point:

▶ Select the **Contour** menu item
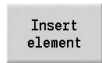
Insert
element

▶ Alternative: Press the **Insert element** soft key

▶ Define the starting point

Parameters for defining the starting point:

- **YS**, **ZS**: **Starting point** of contour
- **WS**: **Starting point** of contour (angle in polar coordinates)
- **PS**: **Starting point** of contour (radius in polar coordinates)

ICP generates a **G180** in **smart.Turn** mode of operation.

Vertical lines in YZ plane

To program a vertical line:

▶ Select the line direction

▶ Enter the line dimensions
▶ Define the transition to the next contour element

Parameters:

- **Y**: **Target point**
- **Yi**: **Target point** (incremental)
- **W**: **Target point** (angle in polar coordinates)
- **P**: **Target point** (in polar coordinates)
- **L**: **Line length**
- **F**: **Further information:** "Machining attributes", Page 438

ICP generates a **G181** in **smart.Turn** mode of operation.
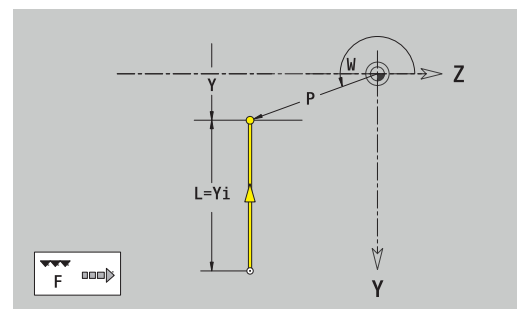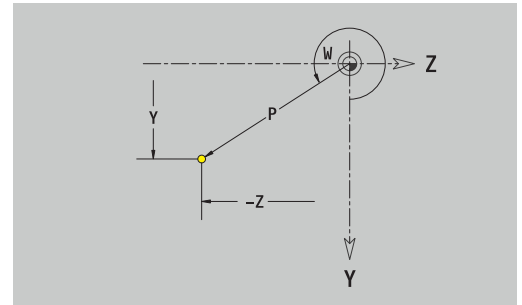
Horizontal lines in YZ plane
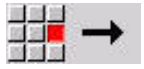
To program a horizontal line:

▶ Select the line direction

▶ Enter the line dimensions
▶ Define the transition to the next contour element

Parameters:

- **Z**: **Target point**
- **Zi**: **Target point** (incremental)
- **W**: **Target point** (angle in polar coordinates)
- **P**: **Target point** (in polar coordinates)
- **L**: **Line length**
- **F**: **Further information:** "Machining attributes", Page 438

ICP generates a **G181** in **smart.Turn** mode of operation.

Line at angle in YZ plane

To program a line at an angle:

▶ Select the line direction

▶ Enter the line dimensions
▶ Define the transition to the next contour element

Parameters:

- **Y**, **Z**: **Target point**
- **Yi**, **Zi**: **Target point** (incremental)
- **W**: **Target point** (angle in polar coordinates)
- **P**: **Target point** (in polar coordinates)
- **L**: **Line length**
- **AN**: **Angle**
- **ANn**: **Angle** to the subsequent element
- **ANp**: **Angle** to the previous element
- **F**: **Further information:** "Machining attributes", Page 438

ICP generates a **G181** in **smart.Turn** mode of operation.
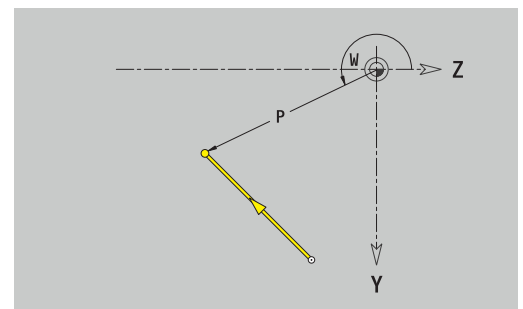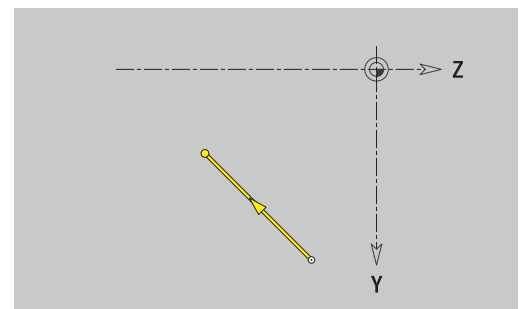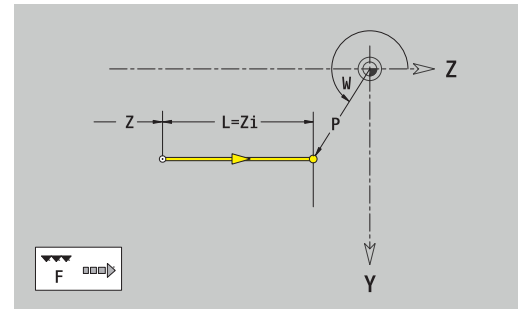
Circular arc in YZ plane

To program a circular arc:

▶ Select the arc's direction of rotation

▶ Enter the arc dimensions

▶ Define the transition to the next contour element

Parameters:
- **Y**, **Z**: **Target point**
- **Yi**, **Zi**: **Target point** (incremental)
- **P**: **Target point** (in polar coordinates)
- **Pi**: **Target point** – Distance between starting point and target point (polar, incremental)
- **W**: **Target point** (angle in polar coordinates)
- **Wi**: **Target point** (polar angle, incremental; reference: starting point)
- **J**, **K**: **Center point** of circular arc
- **Ji**, **Ki**: **Center point** of circular arc (incremental) – Distance between starting point and **Center point** in **Y** and **Z**
- **PM**: **Center point** of circular arc (in polar coordinates)
- **PMi**: **Center point** of circular arc – Distance between starting point and **Center point** (polar, incremental)
- **WM**: **Center point** of circular arc (polar angle)
- **WMi**: **Center point** of circular arc (polar angle, incremental; reference: starting point)
- **R**: **Radius**
- **ANs**: **Angle** – Tangential angle in the starting point
- **ANe**: **Angle** – Tangential angle at the target point
- **ANp**: **Angle** to the previous element
- **ANn**: **Angle** to the subsequent element
- **F**: **Further information:** "Machining attributes", Page 438

ICP generates a **G182** or **G183** in **smart.Turn** mode of operation.
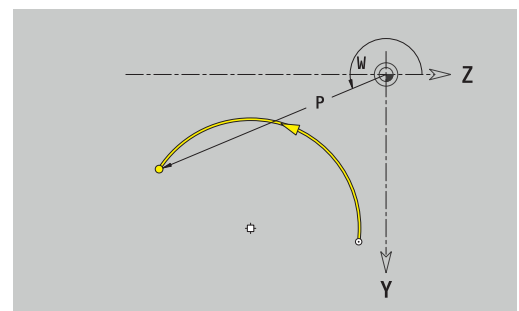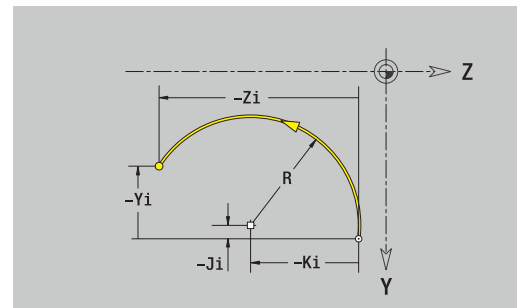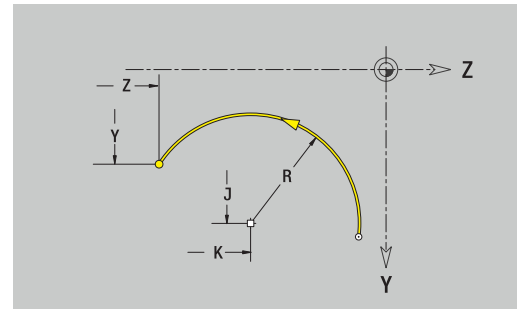
Form elements of YZ plane

Chamfer or rounding in YZ plane

To program a chamfer or rounding radius:

▶ Select the form element menu

▶ Select the Chamfer menu item

▶ Enter the **Chamfer width BR**

▶ Alternative: Select the rounding menu item

▶ Enter the **Rounding radius BR**

▶ Enter the chamfer or rounding as first contour element: **Position AN**

Parameters:

- **BR**: **Cham. width** or **Rounding radius**
- **AN**: **Element pos**
- **F**: **Further information:** "Machining attributes", Page 438

Chamfers or roundings are defined at contour corners. A **contour corner** is the point of intersection between the approaching contour element and the departing contour element. A chamfer or rounding cannot be calculated until the basic contour element is known.

ICP integrates the chamfer or rounding in the **G181**, **G182** or **G183** basic element in **smart.Turn** mode of operation.

Contour begins with a chamfer or a rounding: Enter the position of the **intended corner** as starting point. Then, in the form element menu, select Chamfer or Rounding. Since the **approaching contour element** is missing, enter the **Position AN** to clearly define the position of the chamfer or rounding.

ICP converts a chamfer or rounding at the start of the contour to a linear or circular element.
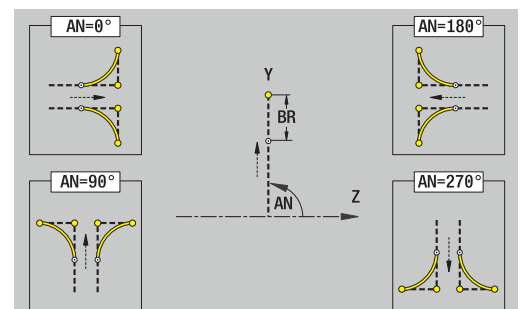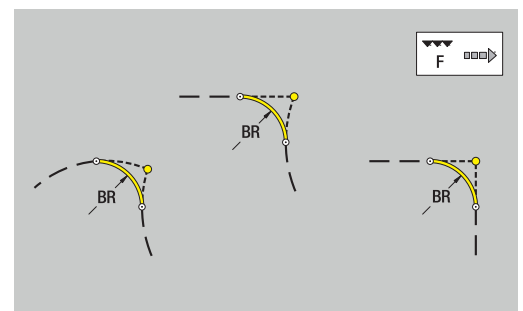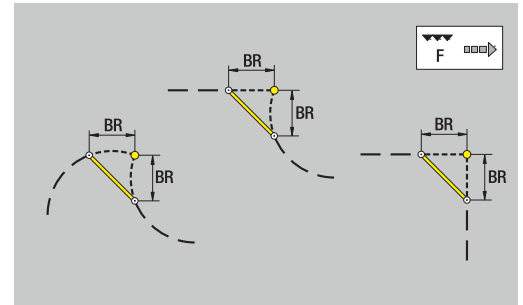
Figures, patterns, and holes in YZ plane (lateral surface)

Circle in YZ plane

Reference data of YZ plane:

- **ID**: **Contour**
- **PT**: **Milling depth**
- **C**: **Spindle angle**
- **XR**: **Reference diameter**

Parameters of figure:

- **YM**, **ZM**: **Center**
- **R**: **Radius**

You can use the **Select reference plane** function to determine the **Reference diameter XR**.

Further information: "Reference data and nested contours", Page 490

ICP generates the following:

- The **SURFACE Y** section code with the **Reference diameter** and **Spindle angle** parameters. The section code is omitted for nested contours.
- A **G308** with the parameters **Contour name** and **Milling depth**.
- A **G384** with the parameters of the figure
- A **G309**

Rectangle in YZ plane

Reference data of YZ plane:

- **ID**: **Contour**
- **PT**: **Milling depth**
- **C**: **Spindle angle**
- **XR**: **Reference diameter**

Parameters of figure:

- **YM**, **ZM**: **Center**
- **A**: **Position angle**
- **K**: **Length**
- **B**: **Width**
- **BR**: **Cham. width** or **Rounding radius**

You can use the **Select reference plane** function to determine the **Reference diameter XR**.

Further information: "Reference data and nested contours", Page 490

ICP generates the following:

- The **SURFACE Y** section code with the **Reference diameter** and **Spindle angle** parameters. The section code is omitted for nested contours.
- A **G308** with the parameters **Contour name** and **Milling depth**.
- A **G385** with the parameters of the figure
- A **G309**

Polygon in YZ plane

Reference data of YZ plane:

- **ID**: **Contour**
- **PT**: **Milling depth**
- **C**: **Spindle angle**
- **XR**: **Reference diameter**

Parameters of figure:

- **YM**, **ZM**: **Center**
- **A**: **Position angle**
- **Q**: **Number of corners**
- **K**: **Width across** – Inscribed circle diameter
- **Ki**: **Edge length**
- **BR**: **Cham. width** or **Rounding radius**

You can use the **Select reference plane** function to determine the **Reference diameter XR**.

Further information: "Reference data and nested contours", Page 490

ICP generates the following:

- The **SURFACE Y** section code with the **Reference diameter** and **Spindle angle** parameters. The section code is omitted for nested contours.
- A **G308** with the parameters **Contour name** and **Milling depth**.
- A **G387** with the parameters of the figure
- A **G309**

Linear slot in YZ plane

Reference data of YZ plane:

- **ID**: **Contour**
- **PT**: **Milling depth**
- **C**: **Spindle angle**
- **XR**: **Reference diameter**

Parameters of figure:

- **YM**, **ZM**: **Center**
- **A**: **Position angle**
- **K**: **Length**
- **B**: **Width**

You can use the **Select reference plane** function to determine the **Reference diameter XR**.

Further information: "Reference data and nested contours", Page 490

ICP generates the following:

- The **SURFACE Y** section code with the **Reference diameter** and **Spindle angle** parameters. The section code is omitted for nested contours.
- A **G308** with the parameters **Contour name** and **Milling depth**.
- A **G381** with the parameters of the figure
- A **G309**
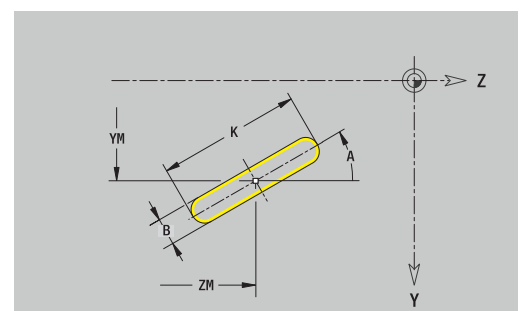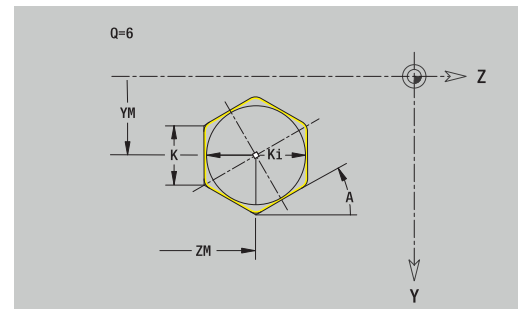
Circular slot in YZ plane

Reference data of YZ plane:

- **ID**: **Contour**
- **PT**: **Milling depth**
- **C**: **Spindle angle**
- **XR**: **Reference diameter**

Parameters of figure:

- **YM**, **ZM**: **Center**
- **A**: **Start angle**
- **W**: **Final angle**
- **R**: **Radius** – Curvature radius (reference: center point path of the slot)
- **Q2**: **Dir.rot.**
    - CW
    - CCW
- **B**: **Width**

You can use the **Select reference plane** function to determine the **Reference diameter XR**.

**Further information:** "Reference data and nested contours", Page 490

ICP generates the following:

- The **SURFACE Y** section code with the **Reference diameter** and **Spindle angle** parameters. The section code is omitted for nested contours.
- A **G308** with the parameters **Contour name** and **Milling depth**
- A **G382** or **G383** with the parameters of the figure
- A **G309**
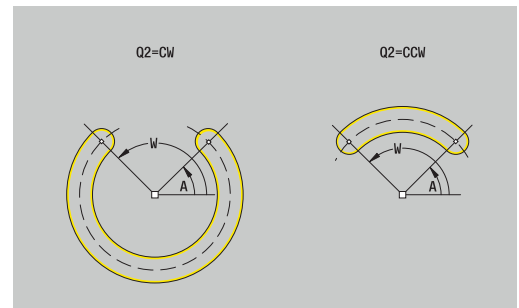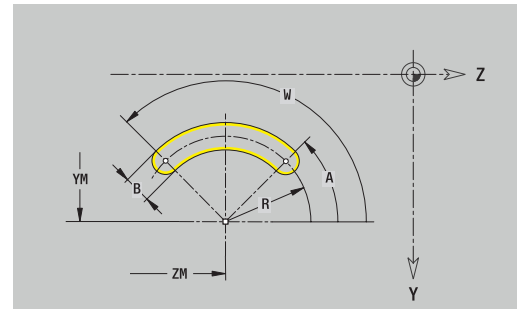
Hole in YZ plane

This function defines a single hole that can contain the following elements:

- **Centering**
- **Hole**
- **Cavity**
- **Thread**

Reference data of hole:

- **ID**: **Contour**
- **C**: **Spindle angle**
- **XR**: **Reference diameter**

Parameters of hole:

- **YM**, **ZM**: **Center**
- **Centering**
  - **O**: **Diameter**
- **Hole**
  - **B**: **Diameter**
  - **BT**: **Depth** (without algebraic sign)
  - **W**: **Angle**
- **Cavity**
  - **R**: **Diameter**
  - **U**: **Depth**
  - **E**: **Sink angle**
- **Thread**
  - **GD**: **Diameter**
  - **GT**: **Depth**
  - **K**: **Cut-in length**
  - **F**: **Thread pitch**
  - **GA**: **Thread direction**
    - **0: Right-hand thread**
    - **1: Left-hand thread**

You can use the **Select reference plane** function to determine the **Reference diameter XR**.

Further information: "Reference data and nested contours", Page 490

ICP generates the following:

- The **SURFACE Y** section code with the **Reference diameter** and **Spindle angle** parameters. The section code is omitted for nested contours.
- A **G308** with the parameters **Contour name** and **Boring depth** (−1*BT)
- A **G380** with the parameters of the hole
- A **G309**
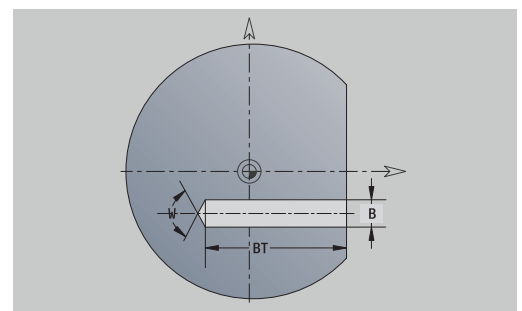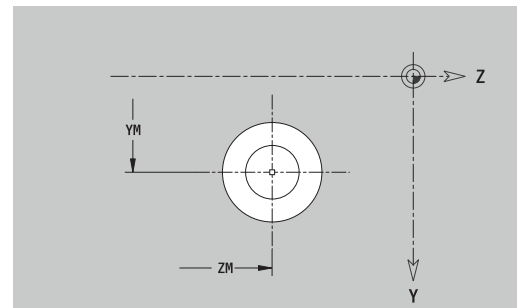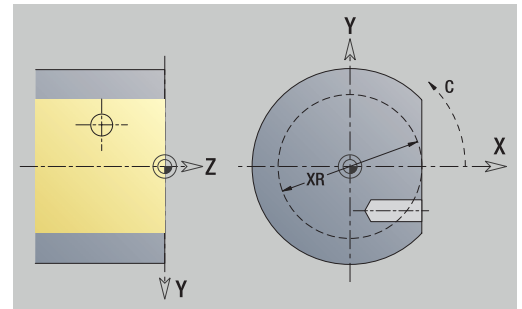
Linear pattern in YZ plane

Reference data of YZ plane:

- **ID**: **Contour**
- **PT**: **Milling depth**
- **C**: **Spindle angle**
- **XR**: **Reference diameter**

Parameters of pattern:

- **Y**, **Z**: **1st pattern point**
- **QP**: **Quantity** of points in pattern
- **JP**, **KP**: **Final point** of pattern (in Cartesian coordinates)
- **JPi**, **KPi**: **Final point** – Distance between two pattern points (in **Y** and **Z**)
- **AP**: **Position angle**
- **RP**: **Length** – Total length of pattern
- **RPi**: **Length** – Distance between two pattern points
- Parameters of selected figure or hole

You can use the **Select reference plane** function to determine the **Reference diameter XR**.

Further information: "Reference data and nested contours", Page 490

ICP generates the following:

- The **SURFACE Y** section code with the **Reference diameter** and **Spindle angle** parameters. The section code is omitted for nested contours.
- A **G308** with the parameters **Contour name** and **Milling depth** or **Boring depth** (−1*BT)
- A **G481** with the parameters of the pattern
- The G code and the parameters of the figure or hole
- A **G309**
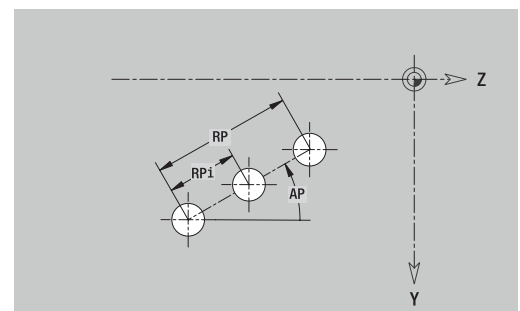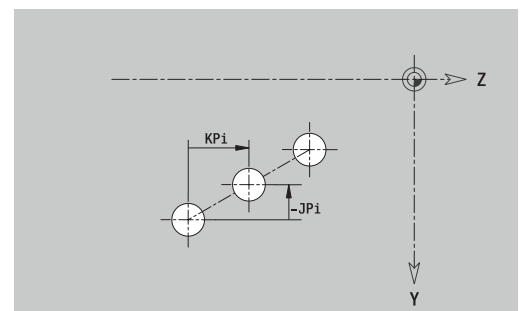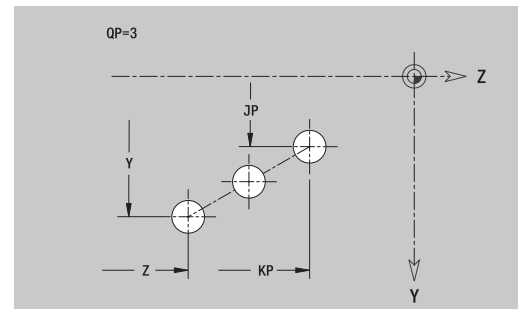
Circular pattern in YZ plane

Reference data of YZ plane:

- **ID**: **Contour**
- **PT**: **Milling depth**
- **C**: **Spindle angle**
- **XR**: **Reference diameter**

Parameters of pattern:

- **Y**, **Z**: **Center** of pattern
- **QP**: **Quantity** of points in pattern
- **DR**: **Dir.rot.** (default: 0)
    - **DR** = 0, without **EP**: Figures are arranged on a full circle
    - **DR** = 0, with **EP**: Figures are arranged on the longer circular arc
    - **DR** = 0, with **EPi**: The algebraic sign of **EPi** defines the direction (**EPi** < 0: clockwise)
    - **DR** = 1, with **EP**: clockwise
    - **DR** = 1, with **EPi**: clockwise (algebraic sign of **EPi** has no effect)
    - **DR** = 2, with **EP**: counterclockwise
    - **DR** = 2, with **EPi**: counterclockwise (algebraic sign of **EPi** has no effect)
- **DP**: **Diameter**
- **AP**: **Start angle** (default: 0°)
- **EP**: **Final angle** (no input: the pattern elements are distributed evenly around 360°)
- **EPi**: **Final angle** – **Angle** between two figures
- **H**: **Element pos**
    - **0: Normal** – The figures are rotated about the circle center (rotation)
    - **1: Master** – The position of the figure relative to the coordinate system remains unchanged (translation)
- Parameters of selected figure or hole

You can use the **Select reference plane** function to determine the **Reference diameter XR**.

ICP generates the following:

- The **SURFACE Y** section code with the **Reference diameter** and **Spindle angle** parameters. The section code is omitted for nested contours.
- A **G308** with the parameters **Contour name** and **Milling depth** or **Boring depth** (−1*BT)
- A **G482** with the parameters of the pattern
- The G code and the parameters of the figure or hole
- A **G309**
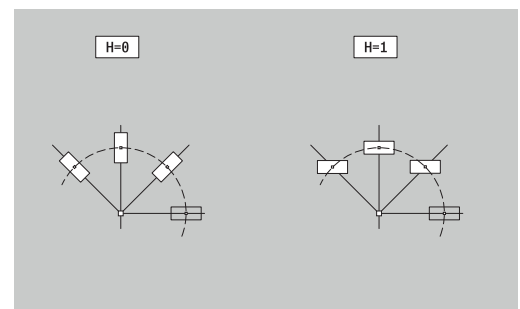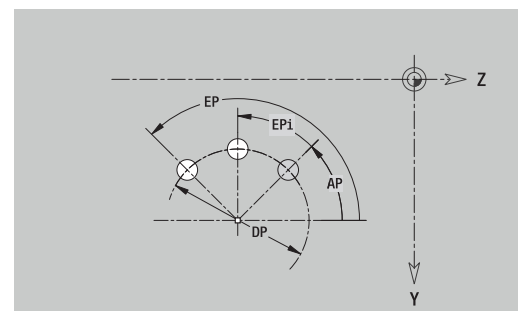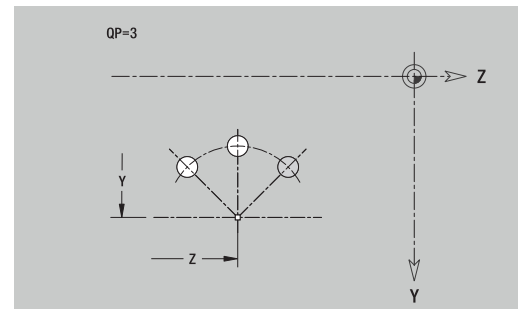
Single surface in YZ plane

This function defines a surface in the YZ plane.

Reference data of YZ plane:

- **ID**: **Contour**
- **C**: **Spindle angle**
- **XR**: **Reference diameter**

Parameters of single surface:

- **Z**: **Reference edge**
- **Ki**: **Depth**
- **K**: **Residual depth**
- **B**: **Width** (reference: **Reference dimension ZR**)
    - **B** < 0: Surface in negative Z direction
    - **B** > 0: Surface in positive Z direction

You can switch between **Depth Ki** and **Residual depth K** by soft key.

You can use the **Select reference plane** function to determine the **Reference diameter XR**.

Further information: "Reference data and nested contours", Page 490

ICP generates the following:

- The **SURFACE Y** section code with the **Reference diameter** and **Spindle angle** parameters. The section code is omitted for nested contours.
- A **G308** with the parameter **Contour name**.
- A **G386** with the parameters of the single surface
- A **G309**

Soft key

Residual depth	Switch the field to the input of **Residual depth K**
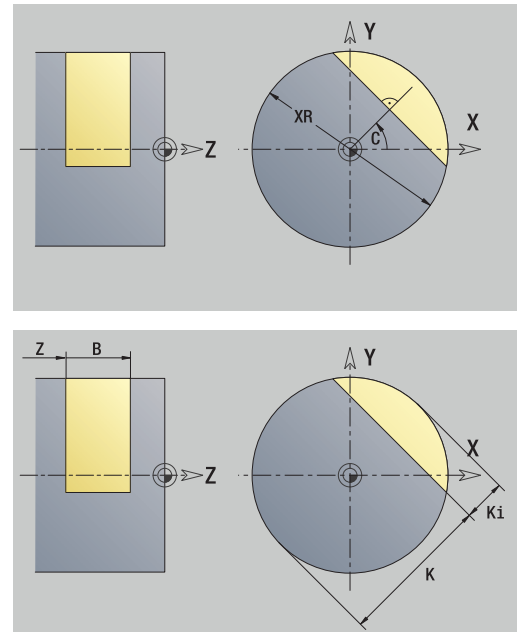
Centric polygon in YZ plane

This function defines centric polygons in the YZ plane.

Reference data of YZ plane:

- **ID**: **Contour**
- **C**: **Spindle angle**
- **XR**: **Reference diameter**

Parameters of single surface:

- **Z**: **Reference edge**
- **Q**: **No. of surfaces** (**Q** >= 2)
- **K**: **Width across** – Inscribed circle diameter
- **Ki**: **Edge length**
- **B**: **Width** (reference: **Reference dimension ZR**)
  - **B** < 0: Surface in negative Z direction
  - **B** > 0: Surface in positive Z direction

You can switch between **Edge length Ki** and **Width across K** by soft key.

You can use the **Select reference plane** function to determine the **Reference diameter XR**.

Further information: "Reference data and nested contours", Page 490

ICP generates the following:

- The **SURFACE Y** section code with the **Reference diameter** and **Spindle angle** parameters. The section code is omitted for nested contours.
- A **G308** with the parameter **Contour name**
- A **G487** with the parameters of the polygon
- A **G309**

Soft key

[icon]	Switch the field to the input of **Width across K**
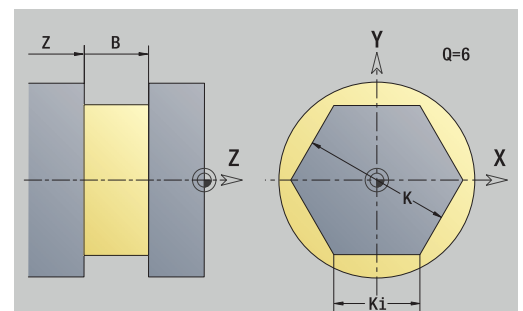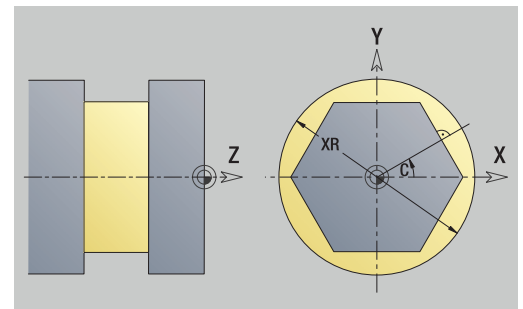
8.16 Loading existing contours

Integrating cycle contours in smart.Turn mode of operation

ICP contours that you have created for **cycle programs** can be loaded in **smart.Turn** operating mode. **ICP** converts the contours into **G** codes and integrates them in the smart.Turn program. The contour is now part of the smart.Turn program.

The **ICP editor** submode considers the type of contour. For example, you can load a contour defined for the face only if you have selected the face (C axis) in **smart.Turn** operating mode.

To integrate a contour:

▶ Activate the **ICP editor** submode
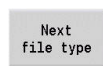
Contour list	▶ Press the **Contour list** soft key
	＞ The **ICP editor** submode opens the **Select ICP contours** window
Next file type	▶ Press the **Next file type** soft key until the cycle contours are displayed
	▶ Select the file
Open	▶ Load the selected file
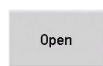
▶ Complete the contour, if necessary
- ■ Workpiece blank or finished part contour: Complete or adapt the contour
- ■ C axis contour: Complete the reference data

Extension	Group
*.gmi	Turning contours
*.gmr	Contours of workpiece blanks
*.gms	Milling contours on front face
*.gmm	Milling contours on lateral surface

DXF contours (option 42)

Contours that exist in DXF format are imported with the **ICP editor**. You can use DXF contours both for **Teach-in** submode and for **smart.Turn** operating mode.

Requirements for the DXF contour:

- Only two-dimensional elements
- The contour must be in a separate layer (without dimension lines, without wraparound edges, etc.)
- Depending on the setup of the lathe, contours for turning operations must be either in front of or behind the workpiece datum (turning center)
- No full circles, no splines, no DXF blocks (macros), etc.

ℹ   The control supports all DXF formats.

Contour preparation during the DXF import: Since the DXF format is fundamentally different from the ICP format, the contour is converted from DXF to ICP format during the import.

Changes during this conversion:

- Polylines are transformed into linear elements
- Gaps between contour elements < 0.01 mm are closed
- Open contours are described from **right to left** (starting point: right)
- The starting point on closed contours is specified according to internal rules
- The rotational direction for closed contours is counterclockwise

Integrating a DXF contour:

▶ Activate the **ICP editor** submode

Contour list	▶ Press the **Contour list** soft key
	> The **ICP editor** submode opens the **Select ICP contours** window.
Next file type	▶ Press the **Next file type** soft key until the DXF contours are displayed (extension: **.dxf**)
	▶ Select the file
	▶ Open the selected file
Open	
Next Contour	▶ Select the DXF layer
Select	▶ Load the selected contour
	▶ Complete the contour, if necessary
	■ Workpiece blank or finished part contour: Complete or adapt the contour
	■ C axis contour: Complete the reference data

8.17 Contour groups

Contour groups in smart.Turn mode of operation

The control supports up to four contour groups in one NC program. The **CONTOUR GROUP** section code initiates the description of a contour group.

You can create a workpiece blank, finished part and auxiliary contours for each contour group. The **ICP editor** submode considers the shift programmed in the specific contour group for the description and display.

G99 assigns the machining operations to a contour group

Display in NC program:

- If you have activated the graphics in the NC program, the control displays the element on which the cursor is positioned when navigating through the contour description
- The control shows the number of the contour group at top left in the graphic window

Display in unit programming:

- If you program an ICP unit in **smart.Turn** operating mode, the control displays **ICP contours**. The various contours and contour groups can be displayed as long as you have not selected a contour in the **FK** parameter

Navigation keys

↓	↑	Switch to the next or previous contour (contour group/workpiece blank/auxiliary contour/finished part)
←	→	Switch to the next contour element

The control shows the number of the contour group and name of the auxiliary contour if applicable at top left in the graphic window.

9

Graphic Simulation

9.1 Simulation submode

Soft key	Meaning
	In **smart.Turn** operating mode, this soft key opens the **Simulation** submode

> **ℹ** The **Simulation** is automatically called in the following operating modes:
> - **Program run** submode
> - **Teach-in** submode
> - **Machine** operating mode (MDI cycles)

When called from **smart.Turn** operating mode, **Simulation** submode opens the large simulation window and loads the selected program. When called from the Machine operating modes, **Simulation** submode opens the small simulation window or the last window you used.
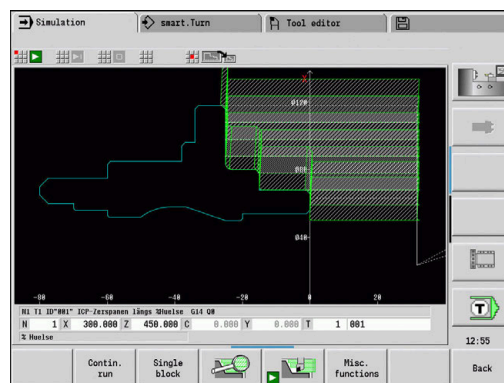
Using Simulation submode

Simulation submode is operated by soft keys in all operating states.

ℹ️ You can operate **Simulation** submode with the menu items (numeric keys), even if the small simulation window is active and the menu bar is not visible.

Starting and stopping with soft keys

▶	Start the simulation from the beginning The soft key switches to the stop symbol; you can now use the soft key to stop and resume the simulation, depending on the simulation status.
▷|	Resume a stopped simulation
◼	Stop the simulation The soft key indicates that a simulation is currently running.

Starting and stopping with menu items

▶	Start the simulation from the beginning
▷|	Resume a stopped simulation
◼	Stop the simulation

Large and small simulation window

Menu item	Meaning
	Switch between the small and the large simulation window This menu item is also effective in the small simulation window, even if the menu bar is not visible.

2-D and 3-D view in smart.Turn mode of operation

Menu item	Meaning
	Switch to 3-D view of the finished workpiece
	Switch to 3-D simulation **Further information:** "3-D simulation in Simulation submode", Page 567
	Switch to 2-D simulation
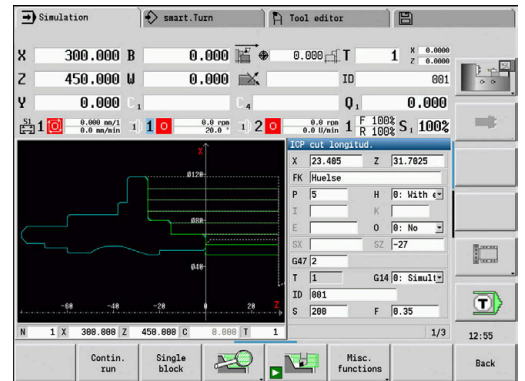
Influencing the simulation speed

Menu item	Meaning
[icon] −	Reduce the simulation speed
[icon] >|<	Activate the programmed feed rate for the simulation speed
[icon] +	Accelerate the simulation speed

Soft keys with active simulation window

Warning no.: 1	Display warnings If the interpreter issues warnings during simulation (e.g., "Residual material due to ..."), the soft key is activated and the number of warnings is shown. Each time the soft key is pressed, it switches to the next warning.
Contin. run	Activate continuous run In **Contin. run** mode, all cycles of the program are simulated in **Program run** submode without stopping.
Single block	Activate Single Block mode In **Single block** mode, the simulation stops after each traverse path (basic block).
[icon]	Open the soft-key menu for the **zoom** function and display the zoom frame **Further information:** "Adjusting the displayed detail", Page 556
Misc. functions	Open the **Misc. functions**
Change variables	Enable the editing of variables during simulation The control activates this soft key only if variables have been defined in the **PROGRAM HEAD**.

With the other menu items and the soft keys listed in the table, you define the sequence of the simulation, activate the zoom function, or use the miscellaneous functions to make settings for the simulation.

> ⓘ
> - In the Machine operating modes, an activation of the **Single block** soft key is also effective in automatic mode
> - In the Machine operating modes, you can start the automatic program run directly from **Simulation** submode with Cycle **On**

The miscellaneous functions

You use the miscellaneous functions to select the simulation window, to determine how the tool path is depicted, or to call the time calculation.

The tables provide you with an overview of the menu and soft-key functions.

Miscellaneous functions menu

	Select the simulation window
	Further information: "Simulation window", Page 549
	Activate a mid-program startup
	Further information: "Simulation with start block", Page 558
	Select the time calculation
	Further information: "Time calculation", Page 560
	Switch between the large and the small simulation window
	Further information: "Using Simulation submode", Page 545
	Switch between the single and multiple window view
	Further information: "Multiple window view", Page 550
	Misc
	■ Save contour
	Further information: "Saving and inserting a contour", Page 561
	■ Dimensioning
	Further information: "Dimensioning", Page 562
	■ Settings
	Further information: "General Settings", Page 564
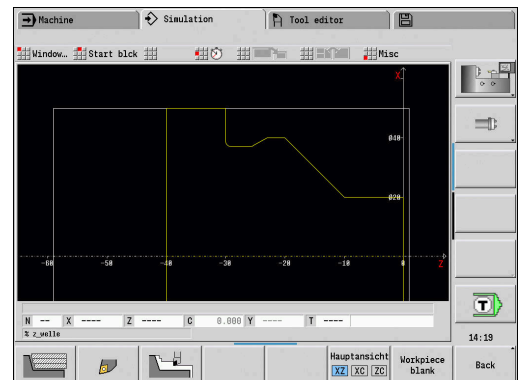
Miscellaneous functions soft keys
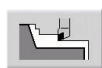
	Switch between wire-frame graphics and cutting-path graphics
	Switch between light-dot and cutting-edge view
	Activate the 2-D material-removal graphic
Next contour group	Switch to the next contour group This soft key is only active if you are working with multiple contour groups.
Main view XZ XC ZC	Select the view
Workpiece blank	Display the internally used workpiece blank for programs without a defined workpiece blank
	Switch the focus to the next window. Only active for multiple window view **Further information:** "Multiple window view", Page 550
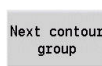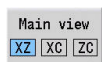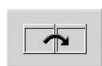
9.2 Simulation window

Setting up the views

With the simulation windows described in the following you check not only the turning operation but also the drilling and milling operations.

- **XZ view (turning view)**: The turning contour is depicted in the XZ coordinate system. The configured coordinate system is taken into account (tool carrier in front of/behind the workpiece, vertical lathes)
- **XC view (face view)**: The displayed coordinate system is a Cartesian system with the axis designations XK (horizontal) and YK (vertical). The angle position C = 0° is on the XK axis, the positive direction of rotation is counterclockwise
- **ZC view (lateral surface view)**: The contour display and traverse-path display are oriented to the position on the **unrolled lateral surface** and the Z coordinates. The upper and lower lines of this **workpiece** correspond to the angular positions C = −180°/+180°, respectively. All drilling and milling operations are within the range −180° to +180°
  - Cycle program or DIN program with workpiece blank definition: The basis for the **unrolled workpiece surface** is the dimensions of the programmed workpiece blank
  - Cycle program or DIN program without workpiece blank definition: The **unrolled workpiece surface** is based on the dimensions of the **standard workpiece blank CfgSimWindowSize** (115200) machine parameter
  - Single cycle or Teach-in: The basis of the **unrolled workpiece surface** is the workpiece section defined by the respective cycle (expansion in Z and **Limit diameter X**)
- **YZ view (side view)**: The contour and the traverse path are shown in the YZ plane. The side view depicts only the Y and Z coordinates—not the spindle position

> ℹ️ Front face and lateral surface windows operate with a **fixed** spindle position. When the lathe rotates the workpiece, **Simulation** submode moves the tool.

Simulation of multiple contour groups

Please note the following when working with multiple contour groups:

- In the **XZ view (turning view)**, the control displays all contour groups.
- In all other views, the control displays the current contour group.
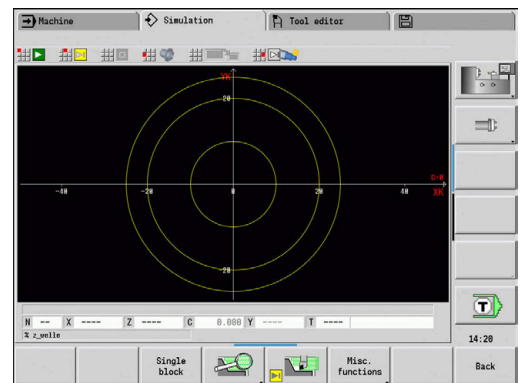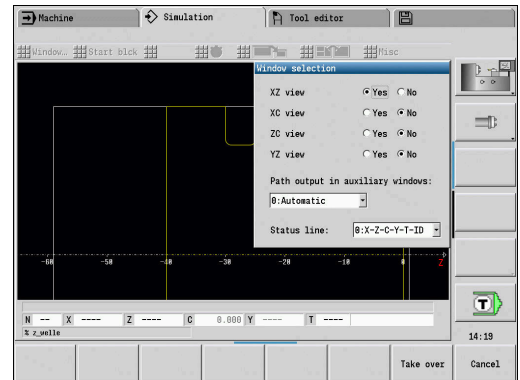
Single-window view

Only one view is shown in the small simulation window. Switch the view using the **Main view** soft key. You can also use this soft key when only one single view is displayed in the large simulation window.

With cycle programs, you can activate the face or lateral surface view only when the C axis is used in the program.

Main view soft key
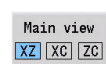
Main view [XZ] [XC] [ZC]	Selecting the view: ■ Turning view XZ ■ Face view XC ■ Lateral surface view ZC

Multiple window view

The multiple window view is possible only in the large simulation window.

To activate the multiple window view:

Misc. functions	▶ Press the **Miscellaneous functions** soft key
[grid icon]	▶ Select the **Window** menu item (in the large simulation window) ▶ Select the desired window combination ▶ Set **Path output in auxiliary windows:**

The path display in the supplementary windows: The front face window, surface window, and YZ view are considered **supplementary windows**.

The following setting specifies whether **Simulation** submode depicts traverse paths in these windows:

■ **Automatic**: **Simulation** submode displays traverse paths if the C axis has been oriented or a **G17** or **G19** has been executed. **G18** or a C axis out of orientation stops the output of the traverse paths

■ **Always**: **Simulation** submode shows every traverse path in all simulation windows

In the multiple window view, a window is marked by a green frame. This window has the **focus**, which means that zoom settings and other functions apply to this window.

To switch the focus:

▶ Press the soft key several times until the focus is on the desired window

▶ Alternative: Press the **GOTO** key

To switch between the single-window and multiple-window view:

▶ Select the menu item (or the decimal point key) to switch from the multiple-window to single-window view

> The window with the green frame is shown as a single-window view

▶ Pressing the menu item again (or the decimal point key) switches back to the multiple-window view

Status display

You can switch the status display in the large simulation window only.

To switch the status display:
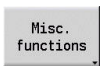
▶ Press the **Miscellaneous functions** soft key

▶ Select the **Window** menu item

▶ Select the desired **Status line:**
- **0: X-Z-C-Y-T-ID** (axis values and tool)
- **1: X-Z-C-Y-G16** (axis values and tilt)
- **2: G95-G96-M-SP** (axis values, feed rate, spindle speed, direction of rotation, and main spindle)

> ⓘ You can also switch the status display with the triple-arrow key in the large simulation window.

9.3 Views

Traverse path display

Rapid traverse paths are shown as a broken white line.

Feed paths are displayed either as a line or as a **cutting path**, depending on the soft-key setting:

- **Line display**: A solid line describes the path of the theoretical tool tip (wire-frame graphics). The wire frame display is particularly convenient if you only need a quick overview of the proportioning of cuts. The path of the theoretical tool tip, however, is not identical with the contour of the workpiece. This view is therefore not as suitable if you wish to run a thorough check of the machined contour. This **falsification** is compensated by the cutting radius compensation
- **Cutting path display**: **Simulation** submode uses hatch marking to depict the surface covered by the **cutting area** of the tool. You can see the area that will actually be machined, with the exact geometry of the cutting edge (cutting radius, cutting width, position of cutting edge, etc.) already accounted for. You can check in **Simulation** submode whether the contour is machined completely or needs to be reworked, whether the contour is damaged by the tool or overlaps are too large. The cutting-path graphics are especially interesting for recessing or drilling operations and for machining inclined surfaces where the tool shape has an essential influence on the accuracy of the resulting workpiece

To activate the cutting path graphics:
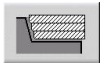
▶ When the soft key is activated, the **cutting paths of traverse** are shown

Tool display

You select by soft key whether the tool cutting edge or the **light dot** is shown:

- The **tool cutting edge** is shown with the correct angles and cutting radius, as defined in the tool database
- **Light dot**: A white square (light dot) is shown at the currently programmed position. The light dot represents the position of the imaginary cutting edge

Miscellaneous functions soft keys
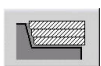
	Switch between wire-frame graphics and cutting-path graphics
	Switch between light-dot and cutting-edge view
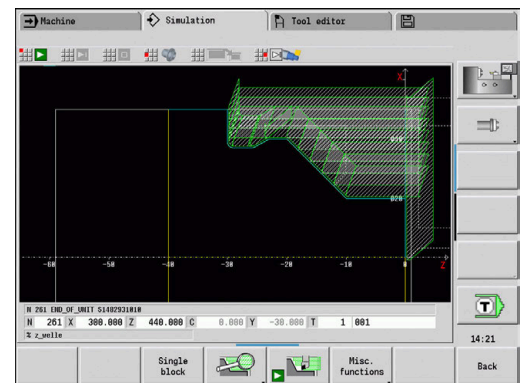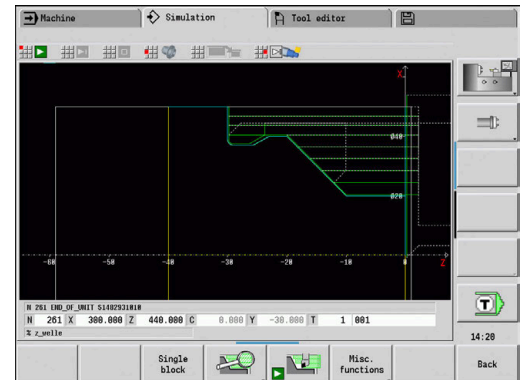
Depicting the tool holder in Simulation submode

The control can depict the associated tool holder with the corresponding dimensions in addition to the tool's cutting edge.

The requirements for this are:

- Creating a new tool holder in the **Holder editor** or selecting an existing holder
- Describing the tool holder with the required parameters (type, dimensions and position)
- The appropriate tool holder must be assigned to the tool (**HID**)

> ⚙ Refer to your machine manual.
>
> The depiction of the tool carrier depends on the machine.
>
> The graphic displays a tool carrier if the following requirements are fulfilled:
>
> - The machine tool builder has saved a description of the tool carrier, e.g. B axis head
> - You have assigned a tool holder to a tool
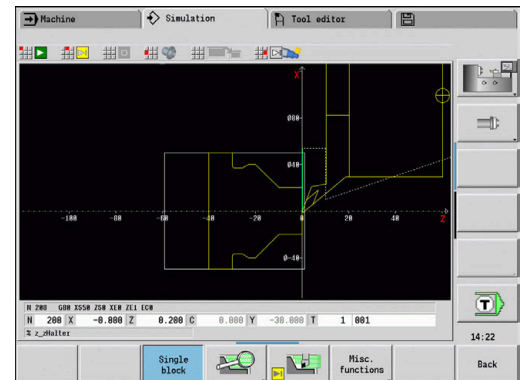
2-D material-removal graphic

2-D material-removal graphic

The 2-D material-removal graphic shows the workpiece blank as a **filled surface**. When the cutting edge passes through the workpiece blank, the workpiece area covered by the tool is erased in the graphic.

The 2-D material-removal graphic mode shows all paths of traverse according to the programmed speed. The 2-D material-removal graphic is only available in turning view (XZ). You activate this type of simulation by soft key.

Miscellaneous functions soft keys

	Activate the 2-D material-removal graphic

3-D view

Soft key	Meaning
	The **3-D view** menu item switches to a perspective view and shows the programmed finished part

With the 3-D view, you can display the workpiece blank and the finished part with all turning operations, milling contours, drilling and boring operations as well as threads in a solid-model view. Tilted Y planes and machining operations referenced to them, such as pockets or patterns, are also displayed correctly by the control.

The control displays milling contours as a function of the **HC**: **Milling/drilling attribute** parameter from **G308**. If you have selected the contour milling, pocket milling or area milling values in this parameter, the graphic shows the respective 3-D elements. If other values are specified in the **HC** parameter or if values are missing, the control depicts the described milling contour as a blue line.

Each element that cannot be calculated by the control is displayed as an orange line; for example, if an open milling contour is programmed as a pocket. With the soft keys and the menu functions, you can influence the display of the workpiece.

3-D view soft keys

	Display the finished part and the programmed workpiece blank
	Display the finished part and the updated workpiece blank
Next contour group	Switch to next contour group This soft key is only active if you are working with multiple contour groups.
	Activate/deactivate the check mode
	Display a section view This setting is also used for the 3-D simulation.
90°	Select the side view Rotate the side view by 90°
	Select the perspective view

> ℹ️ The finished part contour programmed in the **FINISHED** section is shown in the graphic regardless of the machining operation in the NC program.
>
> You can cancel the calculation of the 3-D view by pressing the **ESC** key or the **CANCEL** soft key.
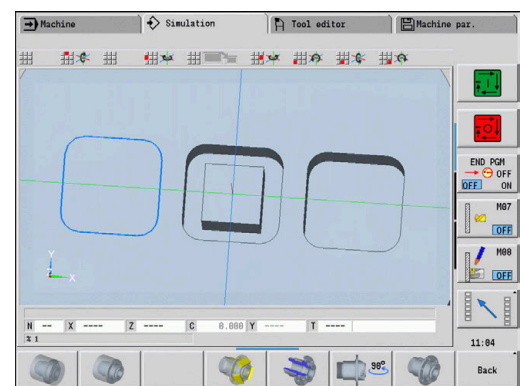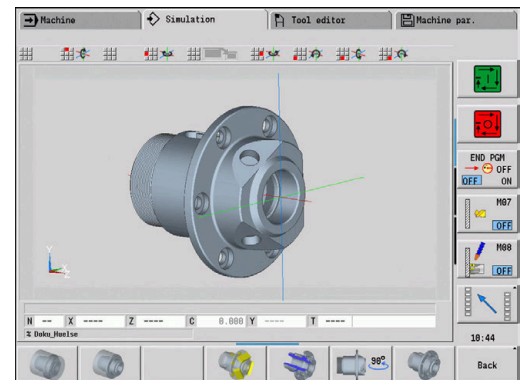
Check mode

The check mode enables you to verify the correct positioning of holes and milling contours, for example.

In the check mode the control displays turning contours in gray, and drilling and milling contours in yellow. For a better overview, the control shows all contours as transparent.
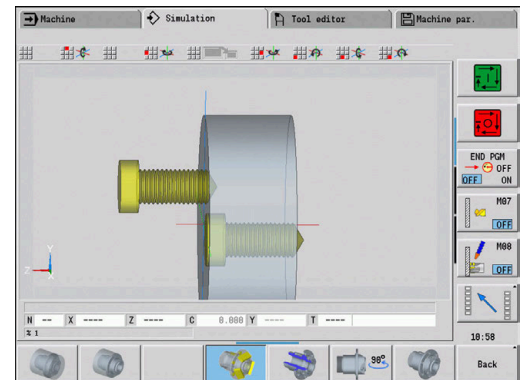
Rotating the 3-D view with the menu functions

With the menu functions you rotate the graphic around the depicted axes. The **perspective view** soft key resets the graphic to its initial condition.

Menu for the 3-D view

	Tilt the graphic backward
	Rotate the graphic horizontally in the indicated direction
	Rotate the graphic horizontally in the indicated direction
	Rotate the graphic counterclockwise
	Tilt the graphic forward
	Rotate the graphic clockwise

Rotating and moving the 3-D view with the mouse

With the right mouse button pressed you can move the displayed workpiece as required.

When you press and hold the left mouse button, you can do the following:

- Vertical movement in the simulation window: Tilt the workpiece forward or backward
- Horizontal movement in the simulation window: Rotate the workpiece horizontally around its own axis
- Horizontal or vertical movement at the edge of the simulation window (gray border): Rotate the workpiece clockwise or counterclockwise
- Movement in any direction: Rotate the workpiece in any direction

9.4 Zooming function in simulation

Adjusting the displayed detail

Soft key	Meaning
[icon]	Press this soft key to activate the **zoom function**

With the zoom menu, you can modify the detail displayed in the simulation window. As an alternative to the soft keys, you can use the arrow keys and the **PgDn** and **PgUp keys** to change the displayed detail.

For cycle programs, and when a program is started for the first time in **Simulation** submode, the control automatically selects the detail. When you simulate the same smart.Turn program again in **Simulation** submode, the last selected detail is used.

In the multiple window view, the zoom function is effective for the window marked by a green frame.

Soft keys in the zoom menu

Soft key	Meaning
Delete paths	▪ Delete all traverse paths drawn so far ▪ If contour follow-up is active, the workpiece blank is updated and newly drawn ▪ Close the zoom menu
Extend view	Extend the displayed detail directly (zoom −)
Zoom off	Switch back to the standard screen detail and close the zoom menu
Last zoom	Return to the last detail selected
Grafik-ausschnitt	Open a window with the values of the current graphics detail. You can copy the values for each 2D main view to a clipboard and paste them in the **G69** function. **Further information**: smart.Turn and DIN Programming User's Manual
Main view XZ XC ZC	Select the view **Further information:** "Setting up the views", Page 549
Take over	Load the area selected by the red frame as the new detail and close the zoom menu
Back	Close the zoom menu without changing the selected detail

Modifying the displayed detail by using keys

You can use the following keys to change the displayed detail without opening the zoom menu.

Keys for modifying the displayed detail

↓ ↑ ← →		The arrow keys move the workpiece in the indicated direction
PG DN		Reduce the displayed workpiece (zoom −)
PG UP		Magnify the displayed workpiece (zoom +)

Modifying the detail with the zoom menu

When you activate the zoom menu, a red frame is shown in the simulation window. This red frame indicates the zoom area that you can select by pressing the **Apply** soft key or the **ENT** key. Use the following keys to change the frame size and position.

Keys for modifying the displayed detail

↓ ↑ ← →		The arrow keys move the red frame in the indicated direction
PG DN		Reduce the size of the red frame
PG UP		Increase the size of the red frame
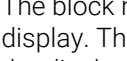
9.5 Simulation with start block

Start block in smart.Turn programs

smart.Turn programs are always simulated from the first block, regardless of which block the cursor is in. If you use the **Start blck** function, **Simulation** submode suppresses all output up to the startup block. If there is a workpiece blank, the simulation scans the blocks up to this position, updates the workpiece blank and redraws it.

The simulation then draws the traverse paths starting from the startup block.

To activate a mid-program startup:

Misc. functions	▶ Press the **Miscellaneous functions** soft key
	▶ Select the **Start blck** menu item
Take over	▶ Enter the number of the startup block
	▶ Transfer the startup block to **Simulation** submode
Back	▶ Go back to the main menu of **Simulation** submode
	▶ Start the simulation
	> The control scans the NC program up to the startup block, updates the workpiece blank and stops at this position
	▶ Continue the simulation

The block number for startup is shown in the lowest line of the display. The field for the startup block and the block number in the display have a yellow background as long as the simulation is performing the block scan.

Mid-program startup remains on even if you cancel the simulation. If you restart the simulation after cancellation, it stops at the **MACHINING** section code. This enables you to change settings before continuing the simulation.

Start blck soft keys

Current line	Load the NC block number in the display as the startup block
Switch off	Switch off the mid-program startup
Take over	Load the defined startup block and activate the mid-program startup
Cancel	Cancel the mid-program startup
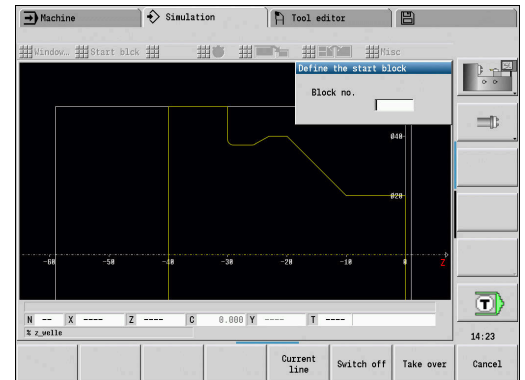
Mid-program startup in cycle programs

For cycle programs, first place the cursor on a cycle and then call **Simulation** submode. The simulation begins with this cycle. All previous cycles are ignored.

The **Start blck** menu item is deactivated for cycle programs.

9.6 Time calculation

Showing the machining times

During simulation, the machining and idle-machine times are calculated. The machining times, idle times and total times are shown in the **time calculation** table (green: machining times; yellow: idle times). If you are working with cycle programs, each cycle is shown in a separate line. In DIN programs, each line represents the use of a new tool (for each tool call with **T**).

If there are more table entries than fit on a screen page, you can call further time data with the arrow keys and **PgUp**/**PgDn**.

To access the machining times:

Misc. functions	▶	Press the **Miscellaneous functions** soft key
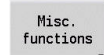
▶ Select the **Time calculation** menu item

9.7 Saving and inserting a contour

Saving the generated contour in Simulation submode

You can save a contour generated in **Simulation** submode and load it into **smart.Turn** operating mode.

Example: Describe the workpiece blank and finished part and simulate machining of the first setup. Then, save the machined contour and use it for the second setup.

During **contour generation** the control saves all contours of the selected contour group.

Simulation submode considers the following shifts of the workpiece datum and/or a mirroring of the workpiece:

- 0: Only move
- 1: Turn over in the main spindle (mirroring)
- 2: Rechuck in the counterspindle (shifting and mirroring)

To save the contour:

Misc. functions
- ▶ Press the **Miscellaneous functions** soft key

- ▶ Select the **Misc** menu item

- ▶ Select the **Save contour** menu item
- ▷ The control opens a dialog box, in which you can define the following input fields:
  - Unit: Contour description metric or in inches
  - Selection of contour group **Q**
  - Type of shift **H**
  - Length of workpiece shift **K**: Shifting the workpiece datum
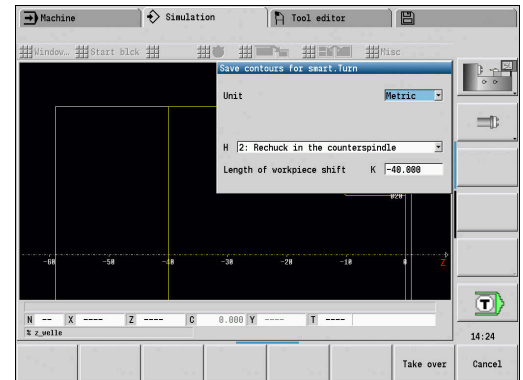
Inserting the saved contour in smart.Turn operating mode

In **smart.Turn** operating mode, you can transfer the workpiece blank contour and the finished part contour generated during simulation to the NC program.

The control provides two options to transfer the saved contours to the NC program:

- Select the **Insert contour** menu item to transfer all of the saved contours.
- Select the **Insert only wkpc blk** menu item to transfer the saved workpiece blank only.

Notes on the **Insert contour** menu item:

- The **Save contour** function in **Simulation** submode converts all contours in all planes of the selected contour group.
- The NC editor replaces all contours during the insertion process. Insert the saved contour only into newly created or copied NC programs, because all previously generated contours will be overwritten. This action cannot be undone.
- The control replaces the contour in which the cursor is positioned only if the NC program contains contour groups.

9.8   Dimensioning

Measuring the generated contour in Simulation submode

You can measure a contour generated in **Simulation** submode or
have the dimensions used during programming displayed.

To measure a contour:

Misc. functions	► Press the **Miscellaneous functions** soft key
	► Select the **Misc** menu item
	► Select the **Dimensioning** menu item
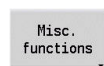
You have the following options:

- Element dimensioning
- Point dimensioning
- Setting a datum

Dimensioning soft keys

	Element forward
	Element back
Next contour group	Select **Next contour group** (only active with multiple contour groups)
Next Contour	Select **Next contour**
Previous contour	Select **Previous contour**

> ⓘ The contour element can also be selected via touch gestures or the mouse.

Element dimensioning menu item

The Element dimensioning menu item is automatically active after you have selected the dimensioning function. All data of the marked contour element are shown in the display below the graphic.

- The arrow indicates the direction of the contour description
- To next contour element: Press the **Element forward/back** soft key
- Change contour: Press the **Previous contour** or **Next contour** soft key

> ⓘ The individual elements are measured for figures.

Point dimensioning menu item

The control displays the dimensions of the contour point relative to the datum.

- To next contour point: Press the **Element forward/back** soft key
- Change contour: Press the **Previous contour** or **Next contour** soft key

Menu item for setting the datum

This function is only available with point dimensioning. It enables you to shift the datum and measure the distance to the datum.

To set a preset:

- ▶ Use the **Element back** soft key to select a new datum
- ▶ Select the **Presetting** menu item
- > The point symbol changes color
- ▶ Use the **Element back** soft key to select the point
- > The control displays the distance relative to the selected datum

Menu item for canceling the preset

To cancel the preset:

- ▶ Select the **Preset from** menu item
- > The preset is canceled.
- > The displayed values once again reference the original preset.

9.9 Settings

General Settings

In **Simulation** submode, you can define general Settings for the simulation.

To define Settings:

[Misc. functions]

▶ Press the **Miscellaneous functions** soft key

▶ Select the **Misc** menu item

▶ Select the **Settings** menu item

The following settings are available:

- Mark the machining area
  Further information: "Mark the machining area", Page 564
- Activate the variable display
  Further information: "Variables", Page 564
- C0 – Marking on workpiece/3D
  Further information: "C0 – Marking on workpiece/3D", Page 565
- Graphic section / Perform G69 function
  Further information: "Graphic section / Perform G69 function", Page 566

Mark the machining area

In 2-D simulation, you can mark the following machining areas:

- 0: Off – No machining area will be marked
- 1: Line – The current cycle will be marked in blue
- 2: Area – The currently machined cycle, including the approach and departure areas will be marked with a blue frame

ℹ️ This setting is volatile.

Variables

Activate the variable display

In 2-D and 3-D simulation, you can display the variables defined in the **PROGRAM HEAD** program section.

Further information: smart.Turn and DIN Programming User's Manual

Activate the variable display

- 0: No – Variables are not displayed
- 1: Yes – Variables are displayed below the simulation window
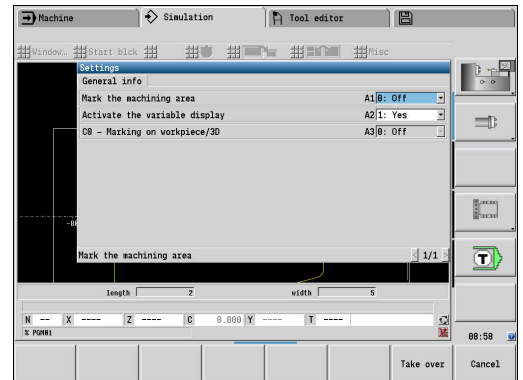
Change variables

In 2-D and 3-D simulation, you can Change variables defined in the **PROGRAM HEAD** program section.

To Change variables:

Change variables	▶ Press the **Change variables** soft key
	> You can now change the variables.
Take over	▶ Press the **Take over** soft key

> 🛈 If you change variables during the Simulation, the Simulation may be aborted.

C0 – Marking on workpiece/3D

In 3-D simulation, you can display the **C0** marking on the workpiece. This way, it is possible to check the position of C axis machining:

- 0: Off – No **C0** marking is displayed
- 1: On – The **C0** marking is shown with a green line

> 🛈 This setting is volatile.

Graphic section / Perform G69 function

The function **G69** allows you to define a graphics detail for the simulation.

Further information: smart.Turn and DIN Programming User's Manual
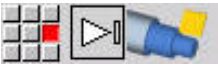
Execute **G69**:

- 0: No – The control ignores **G69** during the simulation.
- 1: Yes – The control executes **G69** during the simulation.

9.10 3-D simulation

3-D simulation in Simulation submode

In **Simulation** submode, you can test a program with the 3-D simulation.

Soft key	Meaning
	The **3-D simulation** menu item activates the 3-D simulation
	The **2-D simulation** menu item deactivates the 3-D simulation

The following functions are identical to 2-D simulation:

- Using the simulation
  Further information: "Using Simulation submode", Page 545
- 3-D view
  Further information: "3-D view", Page 554
- Mid-program startup
  Further information: "Start block in smart.Turn programs", Page 558
- Time calculation
  Further information: "Time calculation", Page 560
- Saving contours
  Further information: "Saving and inserting a contour", Page 561
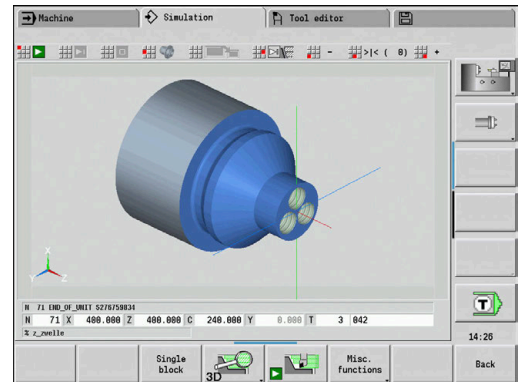
Miscellaneous functions soft keys

	Position a thread pattern on the surface
	Display the workpiece edges
	Display the tool holders

> ℹ️ This function can only be used if values have been specified in the optional **WHT** and **TOF** parameters of the tool holder definition.
> **Further information:** "Holder editor", Page 582

	Switch to the next contour group or switch the focus (green frame) if multiple contour groups are displayed simultaneously
	This soft key is available only if the NC program contains more than one contour group.
	Switch between multiple contour groups and the active contour group
	This soft key is available only if the NC program contains more than one contour group.

Miscellaneous functions soft keys
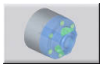
	Switch to the semitransparent workpiece model and back

3-D zoom

The zoom function enables you to show workpiece blanks and finished parts in various perspectives.

To activate the 3D zoom:

▶ Press the **3-D zoom** soft key

> **i** You can rotate the 3-D simulation by using the menu items and the mouse.
>
> **Further information:** "Rotating the 3-D view with the menu functions", Page 555
>
> **Further information:** "Rotating and moving the 3-D view with the mouse", Page 555
>
> If the cutting edge collides with the workpiece during rapid traverse, the sectional planes are displayed in red.

Soft keys for the 3-D zoom

Grafik-ausschnitt	Open a window with the values of the current graphics detail. You can copy the values for the 3D simulation to a clipboard and paste them in the **G69** function. **Further information**: smart.Turn and DIN Programming User's Manual
	Rotate holes or milling contours to the left
	Rotate holes or milling contours to the right
	Select the section view (remains active even during machining)
	Select the side view. Rotate the side view by 90°
	Select the perspective view
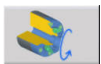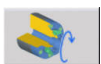
10

Tool and
Technology
Database

10.1 Tool database

You usually program the coordinates for the contour by taking the dimensions from the drawing. To enable the control to calculate the slide path, compensate the cutting radius and determine the proportioning of cuts, you need to enter the tool length, cutting radius, tool angle, etc.

The control can save up to 250 tool data records; each tool data record is identified by an ID number (name). The software option **Tools and Technology** (option 10) enables the control to save up to 999 tool data records. The tool list indicates the maximum number of tool data records and the number of data records found. For each tool, you can enter an additional tool description which makes it easier to find the tool data again when needed.

The **Machine** operating mode provides functions for determining the tool length dimensions.

Further information: "Tool measurement", Page 158

Wear compensation is managed separately. This allows you to enter new compensation values at any time, even during program run.

You can also assign a **cutting material** to the tools, which gives you direct access to the technology database (feed rate, cutting speed). This saves you a lot of time since you only need to determine and enter the cutting data once.
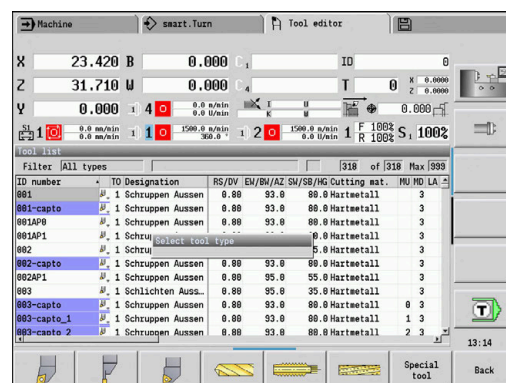
Tool types

Tools for drilling, recessing, finishing, etc., have very different shapes. Therefore, the reference points for determining the tool length and other tool data also vary.

The following table provides an overview of the tool types.

Tool types

Soft key	Function	Description
	Standard turning tools ■ Roughing tools ■ Finishing tools ■ **HDT** tools **Multipoint tool** with tool orientation **TO** = 8	Page 591
	Button tools	Page 591
	Recess. tool ■ Recessing tools ■ Parting tools ■ Recess turning tools	Page 593
	Threading tool	Page 594
	NC center drill	Page 596
	Center drill	Page 597

Tool types

	Counterbore	Page 598
	Countersinker	Page 599
	Standard milling cutter	Page 591
	Twist drill	Page 595
	Indexable-insert drill	Page 595
	Tap	Page 601
	Reamer	Page 600
	Touch probe	Page 607
	Gripper	Page 609
	Thread cutter	Page 603
	Angle cutter	Page 604
	Thin mill cutter	Page 605
	Knurling tool	Page 606
	Stopper tool	Page 608

Multipoint tools

> 🛈 This function is also available on machines with a tool magazine. The control uses the magazine list instead of the turret list.

A multipoint tool is a tool with multiple cutting edges or multiple reference points. A data record is created for every cutting edge or every reference point. Then all data records of the multipoint tool are **linked** together.

Further information: "Editing multipoint tools", Page 578

For every data record of a multipoint tool, the position within the multipoint tool chain is shown in the tool list in the column **MU**. Counting starts from **0**.

In the turret list, multipoint tools are displayed with all cutting edges or reference points. The figure shows a tool with two reference points.
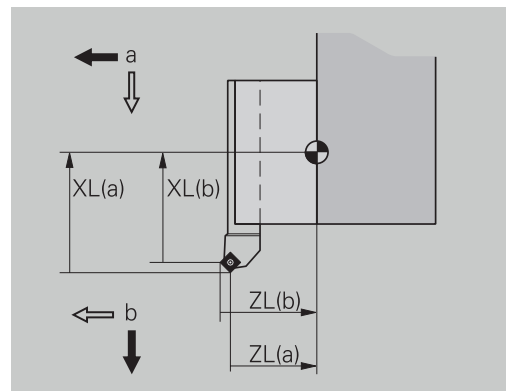
Tool life management

The control can count either the machining time of a tool (i.e. the time a tool is traversed at the programmed feed rate) or the number of parts that were produced with that tool. These two options are used for tool life management.

As soon as the tool life expires or the programmed quantity is reached, the control sets the **diagnostic bit 1**. This causes an error message to be issued the next time the tool is called. If no replacement tool is available, the program run is stopped.

To complete the workpiece currently being produced, press **NC start**.

10.2 Tool editor mode of operation

Navigating in the tool list

In the tool list, the control displays important parameters and the tool descriptions. You can recognize the tool type and the tool orientation from the provided sketch of the tool tip.

You can navigate within the tool list with the arrow keys and **PgUp**/**PgDn** to view the entries. Infrequently used tool parameters are in the list further to the right, and can be displayed by navigating in the columns.

For orientation purposes, the following columns are always displayed:

- ID number
- Tool type
- Tool orientation
- Designation
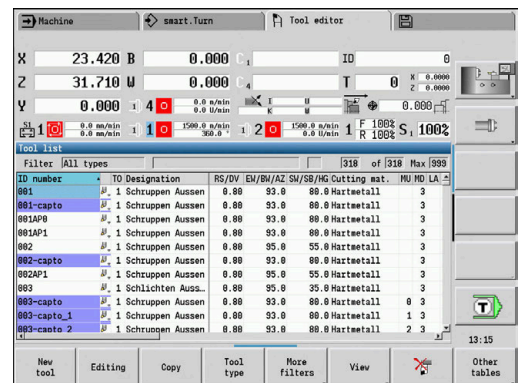
Navigation keys

↓	↑	Switch to the next/previous line (tool) in the tool list
←	→	Switch to the next/previous column in the tool list
PG DN	PG UP	Jump one page up/down in the tool list

> **i** Navigation in the tool list is the same in all modes of operation.

> **i** If any content of a table is incompletely displayed, you can change the width of the table columns.
>
> The changes are effective in all operating modes and are persistent across reboots of the control.

Sorting and filtering the tool list

To display the entries of only one tool type:

| Tool type | ▶ Press the **Tool type** soft key |

▶ Select the tool type in the following soft-key rows
> The control creates a list containing only tools of the selected type

To filter the tool list:

| More filters | ▶ Press the **More filters** soft key |

| Filter orientatn. | ▶ Press the **Filter orientatn.** soft key |

> The control creates a list containing only tools with the selected orientation

| Filter assignment | ▶ Or press the **Filter assignment** soft key |

> The control switches the display between tools in the tool carrier and unassigned tools

| Filter details | ▶ Or press the **Filter details** soft key |

> The control displays a pop-up window containing the available selection criteria
▶ Define the filter criteria

| OK | ▶ Press the **OK** soft key |

To clear filters:

| Filter off | ▶ Press the **Filter off** soft key |

> The control clears the selected filters and displays the complete tool list

To sort the tool list:

| View | ▶ Press the **View** soft key |

| Sort by ID / Type | ▶ Press the **Sort by ID / Type** soft key |

> The tool list switches between **sorting by tool ID number** and **sorting by tool type** (and tool orientation)

| Reverse sorting | ▶ Or press the **Reverse sorting** soft key |

> This toggles the tool list between ascending and descending sorting.

Searching for tools by ID number:
▶ Enter the first few letters or digits of the ID number
> The control jumps to the desired ID number in the open list
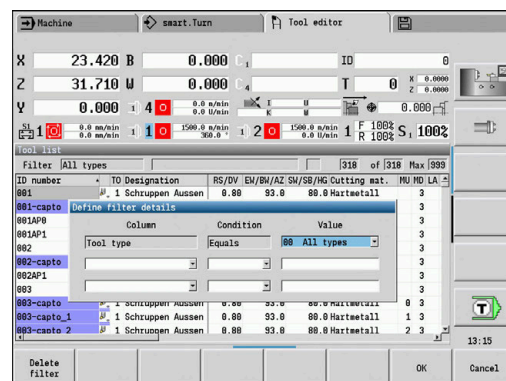
Editing the tool data

To create a new tool:

New tool	▶ Press the **New tool** soft key

- ▶ Select the tool type
- > The control opens the input window
- ▶ Specify the tool orientation
- ▶ Enter further parameters
- ▶ Assign the tool ID number (1 to 16 places, alphanumeric)
- ▶ Assign a tool text

Further information: "Tool texts", Page 577

ⓘ	The control does not show the help graphics for individual parameters until the tool orientation is known.

Tool organization soft keys

New tool	Open the following type selection for creating a new tool

Special tool	Switch to the soft-key row with the special tools

	Type selection for special drilling tools

	Type selection for special milling tools

	Type selection for handling systems and touch probes

Editing	Open the dialog for the selected tool

Copy	Copy the selected tool and create a new tool with the copied data

Tool organization soft keys

[delete icon]	Delete the selected tool from the database after a confirmation request
Technology editor	This soft key is available once you press the **Other tables** soft key. Open **Technology editor** submode. **Further information:** "Technology editor submode", Page 611
Holder editor	This soft key is available once you have pressed the **Other tables** soft key. Open the **Table of the tool holder**.
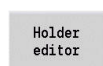
To create new tools by copying:

[↓] ▶ Place the cursor on the desired entry

[Copy] ▶ Press the **Copy** soft key
> The control opens the input window with the tool data
▶ Enter a new tool ID number
▶ Check and edit the other tool data
[Save] ▶ Press the **Save** soft key
> The new tool is added to the database

To edit the tool data:

[↓] ▶ Place the cursor on the desired entry
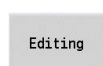
[Editing] ▶ Press the **Editing** soft key
> The tool parameters are provided for editing

To delete an entry:

[↓] ▶ Place the cursor on the desired entry

[delete icon] ▶ Press the **Delete** soft key
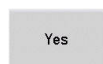
[Yes] ▶ Answer **Yes** to the confirmation prompt
> The control deletes the tool

Tool control graphics

When the tool dialog box is open, the control provides a control graphic you can use to check the specified tools. To do so, press the **Graphic** soft key.

The control generates the displayed tool from the entered parameters. The tool control graphic enables you to check the entered data. Changes become effective as soon as you exit the input field.
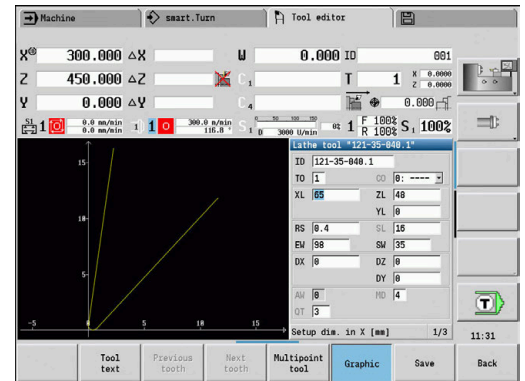
Tool texts

Tool texts are assigned to the tools and displayed in the tool list. The control manages the tool texts in a separate list.

Connections:

- The descriptions are managed in the tool text list. Each entry is preceded by a **QT** number.
- The parameter **Tool text QT** contains the reference number for the **tool text** list. The text indicated by **QT** is then displayed in the tool list

When the tool dialog is open, the control lets you enter tool texts. To do so, press the **Tool texts** soft key.

You can define up to 999 tool texts with a length of max. 80 characters.
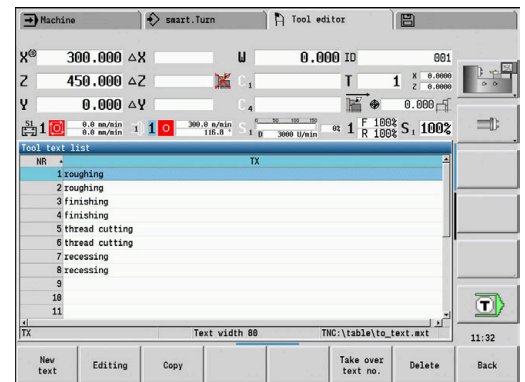
ⓘ ■ New texts are inserted in the next free line after the cursor position
■ When deleting and editing a tool text, please remember that the text might be used for more than one tool

---

Soft keys in the tool text list

New text	Generate a new line in the text list and open it for text input
Editing	Open the selected tool text for editing
Copy	Copy the currently selected tool text to a new text line. This creates a new tool text
Take over text no.	Transfer the text number to the tool dialog as a reference and exit the tool text editor
Save	Save new or edited tool text
Cancel	Discard the current changes
Delete	Delete the selected tool text after a confirmation request
Back	Close the tool text editor and return to the tool dialog without applying any changes to the text references

Editing multipoint tools

To create a multipoint tool:

▶ For each cutting edge, or each reference point, create a separate data record with the tool description

↓	▶ In the tool list, place the cursor on the data record for the first cutting edge
Editing	▶ Press the **Editing** soft key
Multipoint tool	▶ Press the **Multipoint tool** soft key > **Tool editor** operating mode considers this cutting edge to be the **main cutting edge** (**MU** = 0)
↓	▶ Place the cursor on the data record for the next cutting edge
Insert secon. edge	▶ Press the **Insert secon. edge** soft key > **Tool editor** operating mode adds this cutting edge to the multipoint tool sequence
Cuttng edge forward	▶ Select a location for the next cutting edge
	▶ Repeat these steps for any further cutting edges of the multipoint tool
Back	▶ Press the **Back** soft key
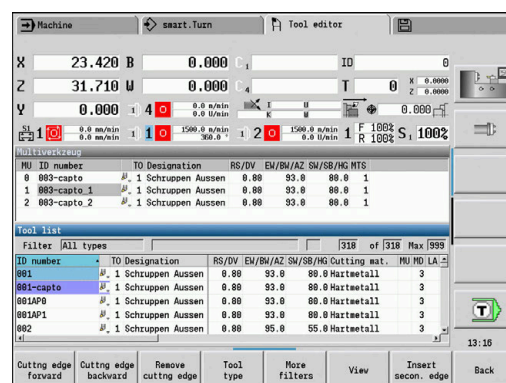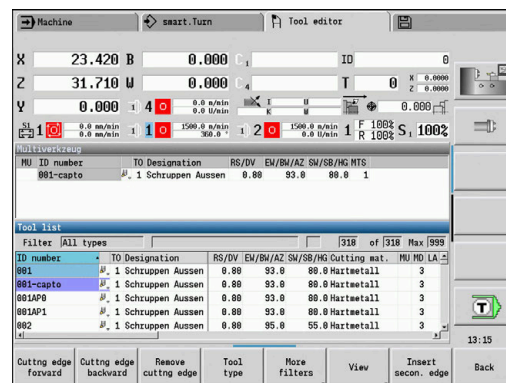
To remove a cutting edge from the multipoint tool:

| ↓ | ▶ | Place the cursor on a cutting edge of the multipoint tool |

| Editing | ▶ | Press the **Editing** soft key |

| Multipoint tool | ▶ | Press the **Multipoint tool** soft key |
| | > | The **Tool editor** operating mode lists all cutting edges of the multipoint tool |

| Cuttng edge forward | ▶ | Select the cutting edge |

| Remove cuttng edge | ▶ | Remove the cutting edge from the multipoint tool chain |

To remove complete multipoint tools:

| ↓ | ▶ | Place the cursor on a cutting edge of the multipoint tool |

| Editing | ▶ | Press the **Editing** soft key |

| Multipoint tool | ▶ | Press the **Multipoint tool** soft key |
| | > | The **Tool editor** operating mode lists all cutting edges of the multipoint tool |

| Cuttng edge forward | ▶ | Place the cursor on the cutting edge no. **0** of the multipoint tool |

| Remove cuttng edge | ▶ | The multipoint tool chain is removed |

Editing tool life data

The control counts the tool life in **RT** and the quantity of finished parts in **RZ**. When the predefined tool life or the part quantity limit has been reached, the tool is considered to be worn out.

To set a tool life limit:

| Dwell |

- ▶ Press the **Dwell** soft key
- > **Tool editor** operating mode opens the **Dwell MT** input field for editing.
- ▶ Enter the tool life of the cutting edge in the **h:mm:ss** format (**h** = hours; **m** = minutes; **s** = seconds). You can use the right and left arrow keys to switch between **h**, **m** and **s**

To enter the quantity:

| No.pieces |

- ▶ Press the **Dwell** soft key
- > The **Dwell** soft key changes to **No.pieces**.
- > **Tool editor** operating mode opens the **No.pieces MZ** input field for editing.
- ▶ Enter the limit quantity, i.e. the number of workpieces that can be machined with one cutting edge

To set a new cutting edge:

- ▶ Insert a new cutting edge
- ▶ Call the associated data record in **Tool editor** operating mode

| New tooth |

- ▶ Press the **New tooth** soft key
- > The tool life or limit quantity is set to **0** and the diagnostic bits are reset.
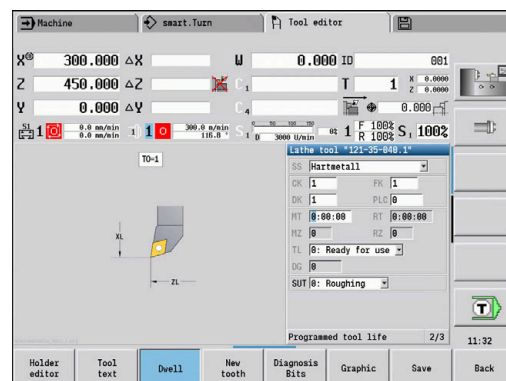
ℹ
- ■ Tool life management can be activated / deactivated in the **lifeTime** (601801) machine parameter
  Further information: "List of machine parameters", Page 620
- ■ The quantity is added when the end of the program has been reached
- ■ Tool life / quantity monitoring is also continued after a change of program

---

Diagnostic bits

The diagnostic bits store information about the status of a tool. The bits are set either by programming in the NC program or automatically by the tool and load monitoring functions.

The following diagnostic bits are available:

- **1 Tool life expired or max. quantity exceeded**
- **2 Breakage detected via load monitoring (limit 2)**
- **3 Wear ascertained via load monitoring (limit 1)**
- **4 Wear according to load monitoring (total load)**
- **5 Wear measured through tool calibration**
- **6 Wear measured via in-process workpiece measurement**
- **7 Wear measured via post-process workpc. measurement**
- **8 Cutting edge is new**
  - New = **1**
  - Used = **0**
- **9 – 15 Not assigned**

If tool life monitoring or quantity monitoring is active and a diagnostic bit has been set, the tool concerned will not be re-inserted during **Program run** submode. If a replacement tool has been defined, it will be inserted instead. If no replacement tool has been defined or if the end of the tool sequence of exchange is reached, the NC program is stopped before the next tool call.

Editing the diagnostic bits

To edit the diagnostic bits in **Tool editor** operating mode:

Editing	▶ Press the **Editing** soft key

Diagnosis Bits	▶ Press the **Diagnosis Bits** soft key
	▶ Use the cursor keys to select the desired bit
GOTO ▢	▶ Press the **GOTO** key to edit the bit

Confirm changes	▶ Save the bit with the Confirm changes soft key
	⊳ The control copies the new diagnostic bits to the **DG** parameter. The data on tool life and quantity limit remain unaffected.

Resetting the diagnostic bits

To reset the diagnostic bits in **Tool editor** operating mode:

Editing	▶ Press the **Editing** soft key

New tooth	▶ Press the **New tooth** soft key

> ⓘ With the **New tooth** soft key, you can reset the diagnostic bits and set bit 8 **Cutting edge is new**. This bit will also be reset as soon as the control inserts the tool.
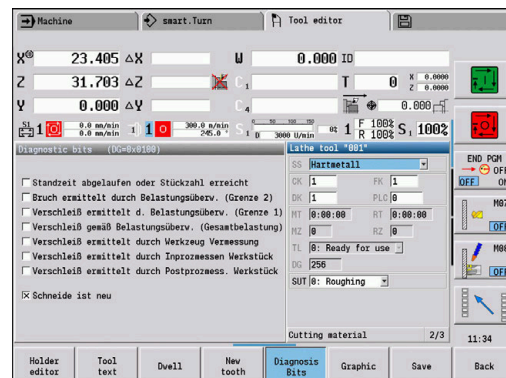
Holder editor

The tool representation in the tool control graphics and in
Simulation submode takes account of the shape of the tool holder
and the mounting position on the tool carrier.

Further information: "Tool control graphics", Page 577

Further information: "3-D simulation in Simulation submode",
Page 567

In the **to_hold.hld** tool holder table, you can define the holder type
and the setting gauges of the tool holder.

To edit the tool holder table in **Tool editor** operating mode:

Other tables	▶ Press the **Other tables** soft key

Holder editor	▶ Press the **Holder editor** soft key

The tool holder table contains the following entries:

- **NR**: **Line number**
- **HID**: **Holder name** – Unique holder name (max. 16 characters)
- **MTS**: **Manual change system**
    - **0: Tool holder**
    - **1: Manual change tool**
- **XLH**: **Setup dim. in X**
- **YLH**: **Setup dim. in Y**
- **ZLH**: **Setup dim. in Z**

- HC: **Holder type**
  - **A1**: Boring bar holder
  - **B1**: right-hand, short design
  - **B2**: left-hand, short design
  - **B3**: right-hand, short design, overhead
  - **B4**: left-hand, short design, overhead
  - **B5**: right-hand, long design
  - **B6**: left-hand, long design
  - **B7**: right-hand, long design, overhead
  - **B8**: left-hand, long design, overhead
  - **C1**: right-hand
  - **C2**: left-hand
  - **C3**: right-hand, overhead
  - **C4**: left-hand, overhead
  - **D1**: Multicarrier
  - **A**: Boring bar holder
  - **B**: Drill holder with coolant supply
  - **C**: Square, longitudinal
  - **D**: Square, transverse
  - **E**: Rear-face machining
  - **E1**: U drill
  - **E2**: Cylindrical shank holder
  - **E3**: Collet chuck holder
  - **F**: Drill holder MK (Morse taper)
  - **K**: Drill chuck
  - **T1**: driven, axial
  - **T2**: driven, radial
  - **T3**: Boring bar holder
  - **X5**: driven, axial
  - **X6**: driven, radial
  - **S2**: all cutting edges of a symmetrical **HDT** tool

    > **i** Data required for 3-D view:
    > - Holder diameter in parameter **WH**
    > - Holder length in parameter **WB**
    >
    > **Further information:** "Example of symmetrical HDT tool", Page 585

  - **S11**: cutting edge 1 of an asymmetrical **HDT** tool
  - **S12**: cutting edge 2 of an asymmetrical **HDT** tool

- **S13**: cutting edge 3 of an asymmetrical **HDT** tool

> **i** Data required for 3-D view:
> - The value defined for the holder diameter is 63 mm and cannot be changed.
> - Value of the differing point angle **SW** in parameter **WH**
> - Holder length in parameter **WB**
>
> **Further information:** "Example of asymmetrical HDT tool", Page 585

- **MP**: **Tool position in turret**
  - **0**: −Z direction
  - **1**: −X/−Z direction
  - **2**: −X/+Z direction
  - **3**: +Z direction
- **WH**: **Height of holder**
- **WB**: **Width of holder**
- **AT**: **Holder type**
- **WHT**: **Depth of holder** (default: parameter **WB**)
- **TOF**: **Offset for depth** (default: parameter **WHT**/2)

> **i** Only ASCII characters are allowed for holder names in the tool holder table. Diacritics or Asian characters are not allowed.
>
> You can also view and edit the tool holder table in open tool forms. Use the **Holder editor** soft key for this purpose.

Soft keys in the tool holder table

New line	Add a new line to the end of the table
Editing	Open the selected tool holder for editing
Copy	Copy the currently selected tool holder to a new text line. This creates a new tool holder
Save	Save the new or edited tool holder
Cancel	Discard the current changes
Delete	Delete the selected tool holder after a confirmation prompt If you logged in with login code 123, the **Delete all** soft key is available. It deletes the entire tool holder table after you confirmed a corresponding prompt and writes this event to the log file.
Back	Close the **Table of the tool holder**.

Example of symmetrical HDT tool

Symmetrical **HDT** tools have three cutting edges that are offset relative to each other by 120°.

For symmetrical HDT tools:

▶ Create each cutting edge separately

Further information: "HDT tools", Page 592

Example values:

- The point angle **SW** of each of the cutting edges is 35°
- Each of the C tilting plane angles **CW** differs from the other by 120° (e.g., 60°, 180°, and 300°)

▶ Link the cutting edges to form a multipoint tool

▶ Assign the holder type **S2** to each cutting edge

▶ Check or, if not present, define the holder diameter in parameter **WH** and the holder length in parameter **WB** as needed
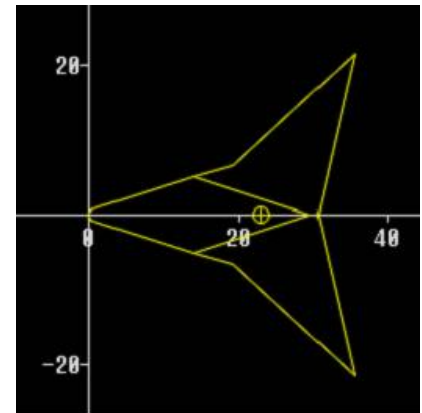
Example of asymmetrical HDT tool

Asymmetrical **HDT** tools have three cutting edges where one cutting edge differs from the other two cutting edges in its point angle. Due to the different point angle, there is also a different angular offset between the cutting edges.

For asymmetrical HDT tools:

▶ Create each cutting edge separately

Further information: "HDT tools", Page 592

Example values:

- The point angle **SW** of the cutting edge 1 is 55°
- The point angle **SW** of each of the cutting edges 2 and 3 is 80°
- Determine and define the C tilting plane angle **CW** as follows:
  - The angle **CW** for the cutting edge 1 results from the actual mechanical position and the tool orientation **TO** (e.g., 180°)
  - The angles **CW** for the cutting edges 2 and 3 have different values, but the same angular offset from the cutting edge 1 (e.g., 67.5° and 292.5°) (**CW** = 180 ±**SW**/2 − **WH**/2)

▶ Link the cutting edges to form a multipoint tool

▶ Assign the appropriate holder type to each cutting edge
  - Holder type **S11** is assigned to cutting edge 1
  - Holder type **S12** is assigned to cutting edge 2
  - Holder type **S13** is assigned to cutting edge 3

▶ Check or, if not present, define the holder length in parameter **WB** as needed

▶ Check or, if not present, define the value of the differing point angle **SW** in parameter **WH** as needed

Example values:

- The parameter **WH** for the cutting edge 1 is 80
- The parameter **WH** for each of the cutting edges 2 and 3 is 55
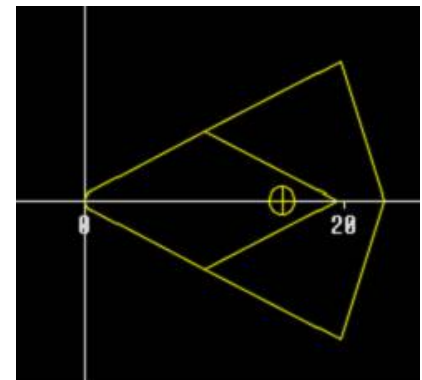
Manual change systems

> ⚙ Refer to your machine manual.
>
> Your machine must be prepared by the machine tool builder if you want to use manual change systems.

A tool holder is designated as a manual change system if it can accommodate various tool inserts via an integral clamping device. Most clamping devices designed as polygon coupling enable rapid, position-precise replacement of tool inserts.

With a manual change system it is possible to insert tools that are not in the turret during a program run. For this, the control checks whether the called tool is in the turret or has to be loaded. If a tool change is required, the control interrupts the program run. After you have manually loaded the tool insert, confirm the tool change and continue the program run.

The following steps are necessary for using manual change systems:

▶ Create the tool holder in the holder table

▶ Select the tool holder in the turret assignment list

▶ Enter the tool data for the manual change tool

Setting up the holder for manual change systems

To set up the manual change system holder in the turret assignment list:

| Turret list | ▶ Press the **Turret list** soft key |

| Special Functions | ▶ Press the **Special functions** soft key |

| Set-up the holder | ▶ Press the **Set-up the holder** soft key |

| Transfer of ID no. | ▶ Press the **Transfer of ID no.** soft key |

> ℹ After setting up a holder for a manual change system in the turret assignment list, three fields of the specific line are color-highlighted.
>
> You can remove a holder for a manual change system with the **Remove tool holder** soft key.
>
> In the turret assignment list, you can only set up the holder type **MTS1** (manual change system). For holder type **MTS0** (standard holder), the control displays an error message.
>
> If the **MTS** parameter of the tool has been set to **1: Manual change tool**, you can define a tool holder. If **0: Tool holder** has been defined, the Set-up the holder soft key is grayed out.

Selecting the manual change system in the tool data

To define the tool as a manual change tool in the tool data form:

| Editing |

▶ Press the **Editing** soft key

▶ On the third form page, select **MTS1: Manual change tool**

| Save |

▶ Press the **Save** soft key

ℹ If you define a tool as a manual change system, in the tool list the tool type field (tool symbol) is color-highlighted.

With manual change tools you must not select a **HID** tool holder (empty field). The assignment between the holder and the tool takes place via the turret assignment list. A manual change system has to have been set up on the corresponding turret pocket for this.

With multipoint tools, all cutting edges must be assigned the same **MTS** input value.

10.3 Tool data

General tool parameters

The parameters listed in the following table are available for all tool types. Parameters for specific tool types are described in the later chapters.

- **ID**: **ID no.** – Tool name (max. 16 characters)
- **TO**: **Tool orientation** (for identification number, see help graphic)
- **XL**: **Setup dim. in X**
- **ZL**: **Setup dim. in Z**
- **DX**: **Wear compensation in X** (range: −10 < **DX** < 10)
- **DZ**: **Wear compensation in Z** (range: −10 < **DZ** < 10)
- **DS**: **Special corr.** (range: -10 < **DS** < 10)
- **MD**: **Rotatnl. dir. M3=3, M4=4** (default: not predefined)
  - 3: **M3**
  - 4: **M4**
- **QT**: Reference to **Tool text**
- **CW**: **C tilting plane angle** – Position of the C axis to determine the work position of the tool (machine-dependent)
- **SS**: **Cutting material** – Name of the cutting material for access to the technology database
- **CK**: **G96 compensation factor** (default: 1)
- **FK**: **G95 compensation factor** (default: 1)
- **DK**: **Depth compensation factor** (default: 1)
- **PLC**: **Additional information**
  Further information: Machine manual
- **MT**: **Programmed tool life** – Default value for tool life management (default: not specified)
- **MZ**: **Programmed quantity** – Default value for tool life management (default: not defined)
- **RT**: **Remaining tool life**
- **RZ**: **Remaining quantity**
- **HID**: **Tool holder designation** – Unique holder name (max. 16 characters)
- **MTS**: **Manual change system**
  - **0: Tool holder**
  - **1: Manual change tool**
- **PTYP**: **Pocket type** (machine-dependent)
- **NMX**: **Max. shaft speed** (spindle speed limitation)

Additional tool parameters and deviating designations in the tool list

> **i** Some of the tool parameters also exist in the turret or magazine assignment list.

- **ID no.**: Content of the **ID** parameter from the tool definition
- **Designation**: Content of the **QT** parameter from the tool definition
- **RS/DV**: Content of the **RS** or **DV** parameter from the tool definition
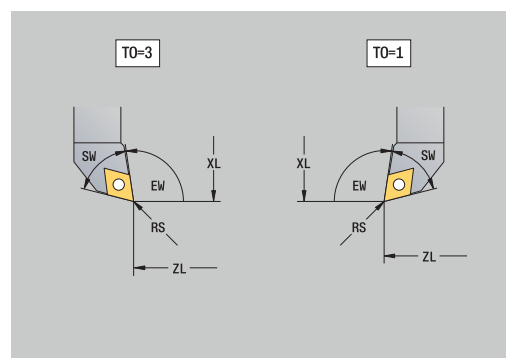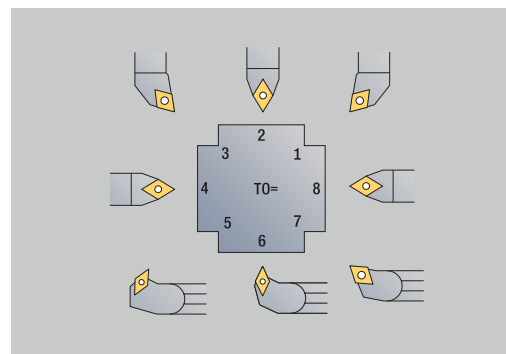
- **EW/BW/AZ**: Content of the **EW**, **BW**, or **AZ** parameter from the tool definition
- **SW/SB/HG**: Content of the **SW**, **SB**, or **HG** parameter from the tool definition
- **Cutting material**: Content of the **SS** parameter from the tool definition
- **MU**: The tool is a **Multipoint tool**
- **LA**: Tool inserted
- **Life**: Remaining tool life / remaining quantity (with tool life monitoring)
- **Status**: With tool life monitoring
- **Diagn.**: Evaluation of diagnostic bits (with tool life monitoring)
- **Holder ID**: Contents of the **HID** parameter from the tool definition

Parameters for drilling tools

- **DV**: **Hole diameter**
- **BW**: **Drill angle** – Point angle of the drill
- **AW**: **Tool driven, no=0/yes=1**
  This parameter specifies, for drilling and tapping tools, whether switching commands will be generated for the spindle or the driven tool during cycle programming.
    - **0**: Stationary tool
    - **1**: Driven tool
- **NL**: **Usable length**
- **RW**: **Position angle** – Deviation from the main machining direction (range: −90° to +90°)
- **AX**: **Salient length in X**
- **FH**: **Height of the chuck for driven tool**
- **FD**: **Diameter of the chuck**

Description of the tool parameters

- **ID number** (**ID**): The control needs a unique name for each tool. This **ID number** can have up to 16 alphanumeric characters.
- **Tool orientation** (**TO**): From the tool orientation, the control determines the position of the tool tip and, depending on the selected tool type, additional information such as the tool angle direction, position of the tool reference point, etc. This information is necessary, for example, for calculating the cutting radius compensation, plunge angle, etc.
- **Setup dimensions** (**XL**, **ZL**): They refer to the tool reference point. The position of the reference point depends on the tool type (see help graphics)

- **Compensation values** (**DX**, **DZ**, **DS**): They compensate for the wear of the cutting edge. For recessing and button tools, **DS** stands for the compensation value of the third side of the tool, which is away from the tool reference point. The compensation values have four decimal places for **mm** units and five decimal places for **inch** units. The cycles automatically switch to the special compensation. With **G148**, you can also switch to special compensation for single-path machining operations
- **Direction of rotation** (**MD**): If you define a direction of rotation, in the cycles using this tool a switching command (**M3** or **M4**) is automatically generated either for the spindle or, with driven tools, for the auxiliary spindle

    > It depends on the PLC software of your machine whether the generated switching commands are evaluated. If the PLC does not execute the switching commands, this parameter should not be entered. Refer to your machine manual.

- **Tool text** (**QT**): You can assign a tool text to each tool. The text is shown in the tool lists. Since the tool texts are maintained in a separate list, you can enter the reference to the text in **QT**
    Further information: "Tool texts", Page 577
- **Cutting material** (**SS**): This parameter is required if you want to use the cutting data from the technology database
    Further information: "Technology database", Page 610
- **Compensation factors** (**CK**, **FK**, **DK**): These parameters serve for the tool-specific adjustment of the cutting values. The cutting data from the technology database are multiplied by the compensation factors before they are entered as default values
- **Additional information** (**PLC**): You can find information on this parameter in the machine manual. This parameter can be used for machine-specific settings.
- **Idle time** (**MT**, **RT**): If you use the tool life management, specify the tool life of the cutting edge in **MT**. In **RT**, the control shows the tool life already **passed**.
- **No.pieces** (**MZ**, **RZ**): With tool life management, use **MZ** to specify the number of workpieces that can be machined with one cutting edge. In **RZ** the control shows the number of workpieces that have already been machined with this cutting edge.

    > The tool life monitoring and the quantity count are used alternatively.

- **Manual change system** (**MTS**): Define the tool holder

Standard turning tools

New tool	▶ Press the **New tool** soft key
	▶ Press the **Turning tool** soft key
	▶ Alternative: For tools with round indexable insert, switch to the dialog for **Button tools**

The tool orientations **TO** = **1, 3, 5, and 7** allow you to enter the **Setting angle EW**. The tool orientation values **TO** = **2, 4, 6, and 8** are used for neutral tools. **Neutral** tools are tools that are centered precisely in the tool tip. One of the setup dimensions for neutral tools is referenced to the center of the tool-tip radius.

Special parameters for roughing and finishing tools:

- **CO**: **Cutting edge position**
  The main machining direction of the tool influences the orientation of the tool angle **EW** and the point angle **SW** (required for **AWG** submode with **TURN PLUS**).
  - **1: Longitudinal preferred**
  - **2: Transverse preferred**
  - **3: Only longitudinal**
  - **4: Only transverse**
- **RS**: **Cutting edge radius**
- **EW**: **Setting angle** (range: 0° <= **EW** <= 180°)
- **SW**: **Point angle** (range: 0° <= **SW** <= 180°)
- **SUT**: **Tool type** (required for **AWG** submode in **TURN PLUS**)
- Further tool parameters:
  Further information: "General tool parameters", Page 588

Special parameters for button tools:

- **RS**: **Cutting edge radius**
- **EW**: **Setting angle** (range: 0° <= **EW** <= 180°)
- **DS**: **Special corr.** (position of the special compensation; see figure)
- **NL**: **Usable length** (for neutral button tools)
- Further tool parameters:
  Further information: "General tool parameters", Page 588

> ℹ The **DX**, **DZ wear compensation** compensates for wear on the two sides of the cutting edge that lie next to the tool reference point. The **Special corr. DS** compensates for wear on the third side of the cutting edge.
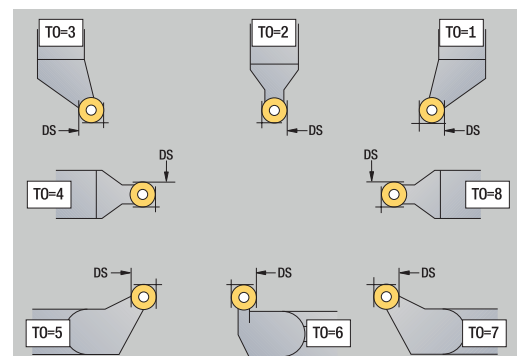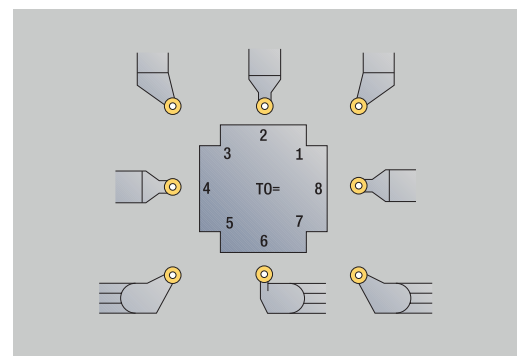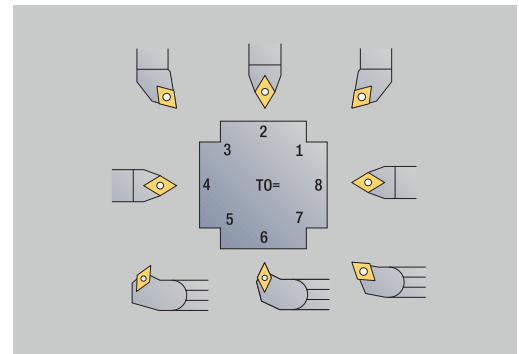
HDT tools

You can use all neutral standard turning tools with the tool orientation **TO** = 8 and the cutting-edge insert position **CO** = 0 as **HDT** tools.

If you would like to use a tool in **HDT** mode, then define the parameter **HD** = 1.

You define the parameters **XL**, **YL**, **ZL**, and **CW** based on the selected tool orientation relative to a B-axis angle of 0°.

Because the standard **HDT** tools consist of three individual cutting edges, you need to link the desired tool data to form a multipoint tool.

To ensure that the **HDT** tool is correctly depicted in the simulation, you also need to assign the correct tool holders to the individual cutting edges.

Further information: "Holder editor", Page 582

Notes:

- The center of rotation of the multipoint tool lies at the point of intersection of the angle bisectors of all individual cutting edges.
- To define the parameter **XL**, you always use a negative value; to define the parameter **YL**, you use a value of 0.
- The parameter **CW** may vary depending on the point angles of the individual cutting edges. The angular offset between the cutting edges is constant for symmetrical **HDT** tools; it is not constant for asymmetrical **HDT** tools.

  Further information: "Example of asymmetrical HDT tool", Page 585

- The parameter **DN** allows you to define a safety clearance. Its effect corresponds to the effect of parameter **SL** of the simultaneous turning cycle. The safety clearance is shown in the 2-D simulation.
- The parameter **SD** allows you to define the tool length for the 3-D simulation.
- **HDT** tools are not used in automatic working plan generation (AWG).

Recessing tools

 ▶ Press the **New tool** soft key

 ▶ Press the **Recessing tool** soft key

Recessing tools are used for recessing, parting, recess turning and finishing (only in the **smart.Turn** mode of operation).

Special parameters for recessing tools:

- **RS**: **Cutting edge radius**
- **SW**: **Point angle**
- **SB**: **Cutting width**
- **SL**: **Cutting length**
- **DS**: **Special corr.**
- **SUT**: **Tool type** (required for **AWG** submode in **TURN PLUS**)
  - **0: Recessing**
  - **1: Parting**
  - **2: Rec. turn.**
- **DN**: **Tool width**
- **SD**: **Shank diameter**
- **ET**: **Max. plunging depth**
- **NL**: **Usable length**
- **RW**: **Angular offset** (only for B axis)
- Further tool parameters:
  Further information: "General tool parameters", Page 588

ℹ️ The **DX**, **DZ wear compensation** compensates for wear on the two sides of the cutting edge that lie next to the tool reference point. The **Special corr. DS** compensates for wear on the third side of the cutting edge.
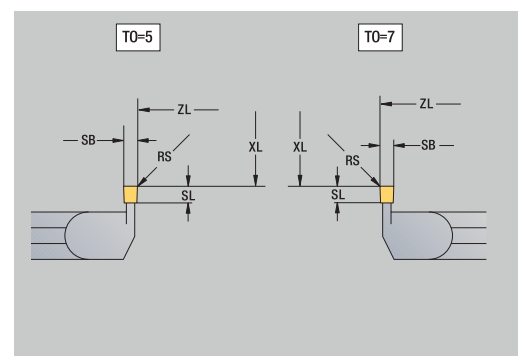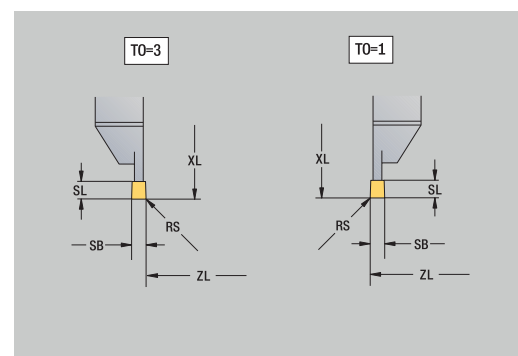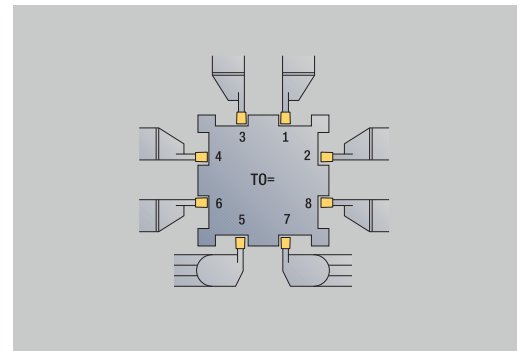
Thread-cutting tools

▶ Press the **New tool** soft key

▶ Press the **Threading tool** soft key

The help graphics illustrate the dimensions of the tools.

Special parameters for thread cutting tools:

- **RS**: **Cutting edge radius**
- **SB**: **Cutting width**
- **EW**: **Setting angle** (range: 0° <= **EW** <= 180°)
- **SW**: **Point angle** (range: 0° <= **SW** <= 180°)
- **DN**: **Tool width**
- **SD**: **Shank diameter**
- **ET**: **Max. plunging depth**
- **NL**: **Usable length**
- Further tool parameters:
  Further information: "General tool parameters", Page 588
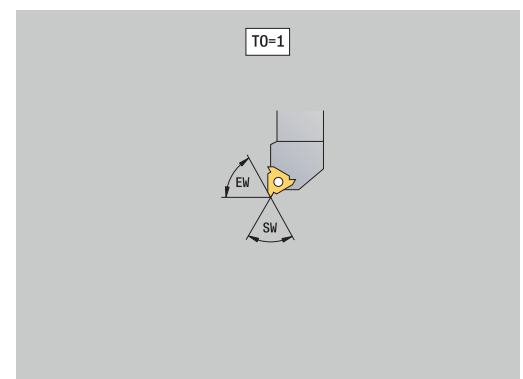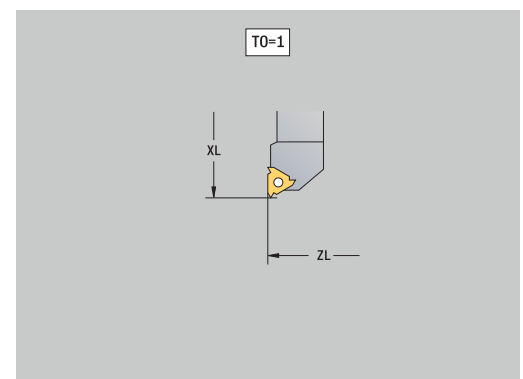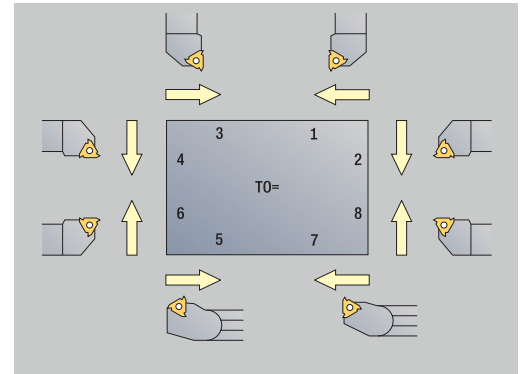
Twist drills and indexable-insert drills

New tool	▶ Press the **New tool** soft key
	▶ Press the **Drill** soft key
	▶ Alternative: For indexable-insert drills, switch to the dialog for **Indexable-insert drill**

The help graphics illustrate the dimensions of the tools.

Special parameters for twist drills:

- **DV**: **Hole diameter**
- **BW**: **Drill angle** – Point angle of the drill
- **AW**: **Tool driven, no=0/yes=1**
  This parameter specifies, for drilling and tapping tools, whether switching commands will be generated for the spindle or the driven tool during cycle programming.
  - **0**: Stationary tool
  - **1**: Driven tool
- **NL**: **Usable length**
- **RW**: **Position angle** – Deviation from the main machining direction (range: −90° to +90°)
- **AX**: **Salient length in X**
- **FH**: **Height of the chuck for driven tool**
- **FD**: **Diameter of the chuck**
- Further tool parameters:
  Further information: "General tool parameters", Page 588

> ℹ For drilling operations at **constant cutting speed**, the **Hole diameter DV** parameter is used to calculate the spindle speed.

NC center drill
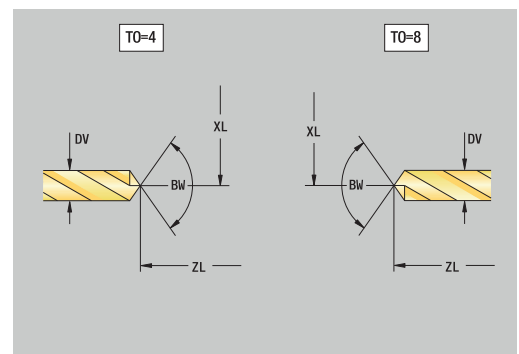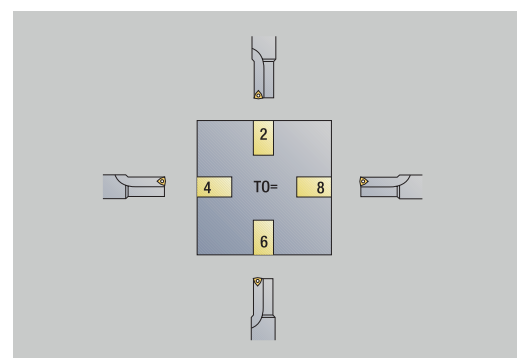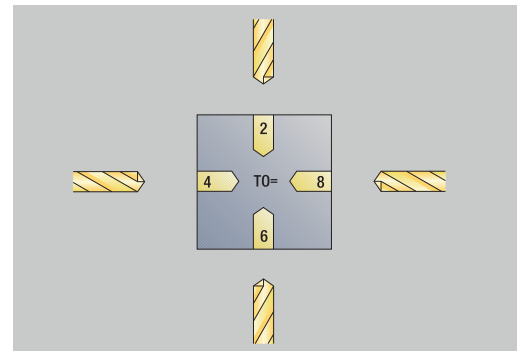
New tool	▶ Press the **New tool** soft key
Special tool	▶ Press the **Special tool** soft key
[drill icon]	▶ Press the **special drilling tools** soft key
[drill icon]	▶ Press the **NC center drill** soft key

The help graphics illustrate the dimensions of the tools.

Special parameters for NC center drills:

- **DV**: **Hole diameter**
- **BW**: **Drill angle** – Point angle of the drill
- Further tool parameters:
  Further information: "General tool parameters", Page 588

> 🛈 For drilling operations at **constant cutting speed**, the **Hole diameter DV** parameter is used to calculate the spindle speed.
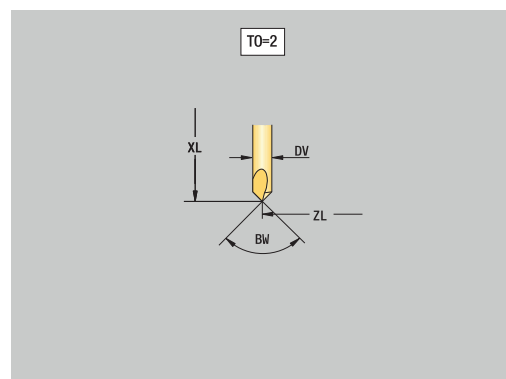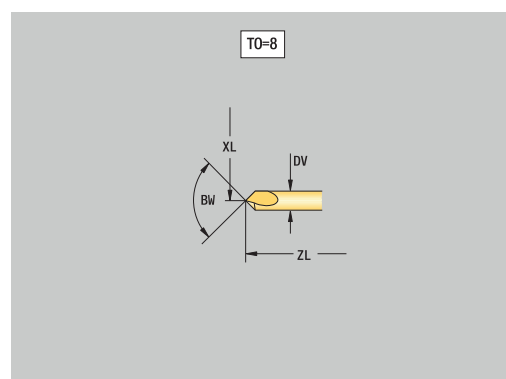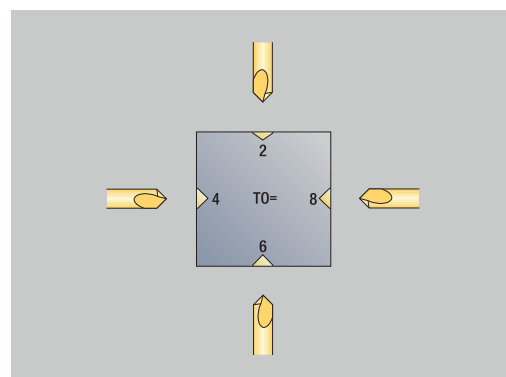
Centering tool

| | ▶ Press the **New tool** soft key |

| | ▶ Press the **Special tool** soft key |

| | ▶ Press the **special drilling tools** soft key |

| | ▶ Press the **Cent. drill** soft key |

The help graphics illustrate the dimensions of the tools.

Special parameters for centering tools:

- **DV**: **Hole diameter**
- **DH**: **Stud diameter**
- **BW**: **Drill angle** – Point angle of the drill
- **SW**: **Point angle**
- **ZA**: **Stud length**
- Further tool parameters:
  Further information: "General tool parameters", Page 588

> ℹ️ For drilling operations at **constant cutting speed**, the **Hole diameter DV** parameter is used to calculate the spindle speed.

Counterbore
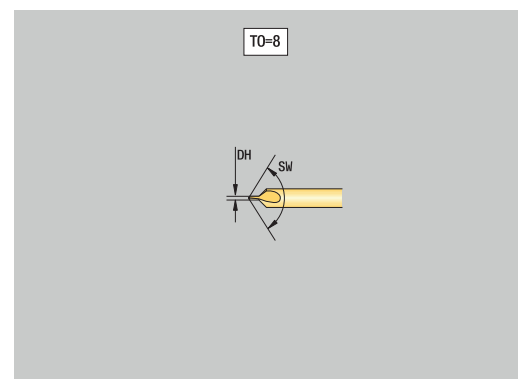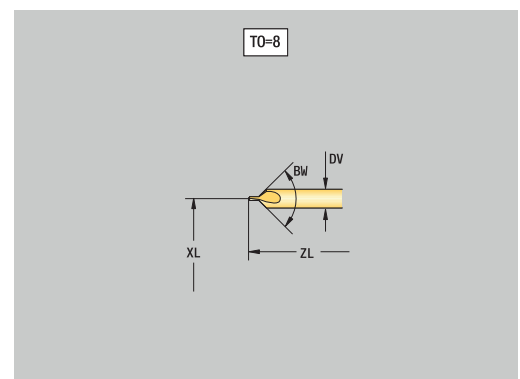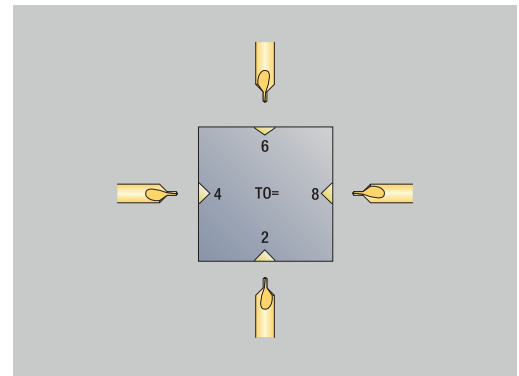
New tool	▶ Press the **New tool** soft key
Special tool	▶ Press the **Special tool** soft key
	▶ Press the **special drilling tools** soft key
	▶ Press the **Counterbore** soft key

The help graphics illustrate the dimensions of the tools.

Special parameters for counterbores:

- **DV**: **Hole diameter**
- **DH**: **Stud diameter**
- **ZA**: **Stud length**
- Further tool parameters:
  Further information: "General tool parameters", Page 588

> ℹ For drilling operations at **constant cutting speed**, the **Hole diameter DV** parameter is used to calculate the spindle speed.

Countersink

New tool	▶ Press the **New tool** soft key
Special tool	▶ Press the **Special tool** soft key
(drill icon)	▶ Press the **special drilling tools** soft key
(countersink icon)	▶ Press the **Countersinker** soft key

The help graphics illustrate the dimensions of the tools.

Special parameters for countersinks:

- **DV**: **Hole diameter**
- **DH**: **Stud diameter**
- **BW**: **Drill angle**
- Further tool parameters:
  Further information: "General tool parameters", Page 588

ℹ For drilling operations at **constant cutting speed**, the **Hole diameter DV** parameter is used to calculate the spindle speed.

Reamer
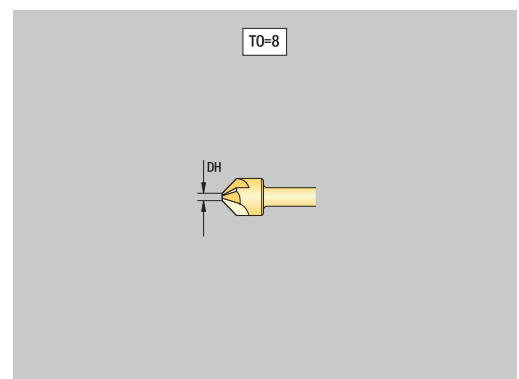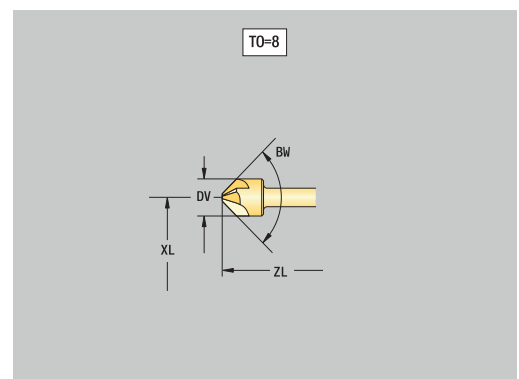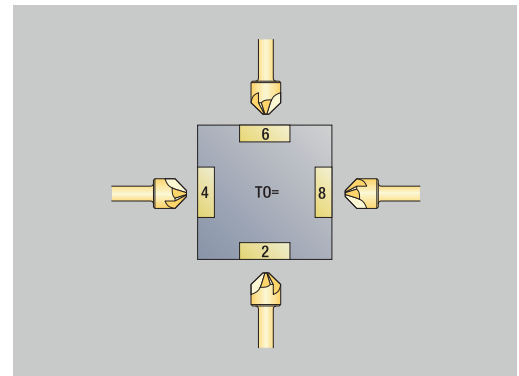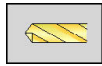
New tool	▶ Press the **New tool** soft key
Special tool	▶ Press the **Special tool** soft key
	▶ Press the **special drilling tools** soft key
	▶ Press the **Reamer** soft key

The help graphics illustrate the dimensions of the tools.

Special parameters for reamers:

- **DV**: **Hole diameter**
- **DH**: **Stud diameter**
- **AL**: **Cut-in length**
- Further tool parameters:
  Further information: "General tool parameters", Page 588

> 🖝 For drilling operations at **constant cutting speed**, the **Hole diameter DV** parameter is used to calculate the spindle speed.

Tap

[New tool]

▶ Press the **New tool** soft key

[Tap]
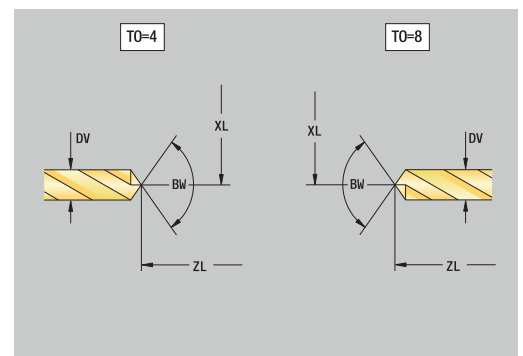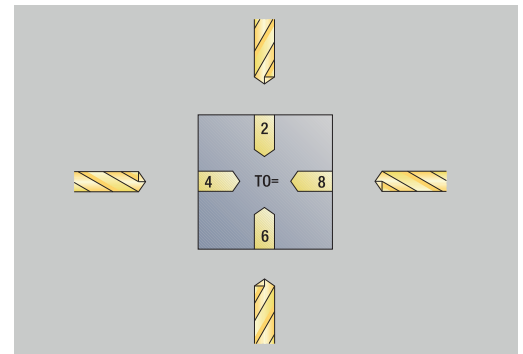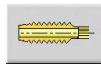
▶ Press the **Tap** soft key

The help graphics illustrate the dimensions of the tools.

Special parameters for taps:

- **DV**: **Thread diameter**
- **HG**: **Thread pitch**
- **AL**: **Cut-in length**
- Further tool parameters:
  Further information: "General tool parameters", Page 588

ℹ The **Thread pitch HG** is evaluated if the corresponding parameter has not been defined in the tapping cycle.
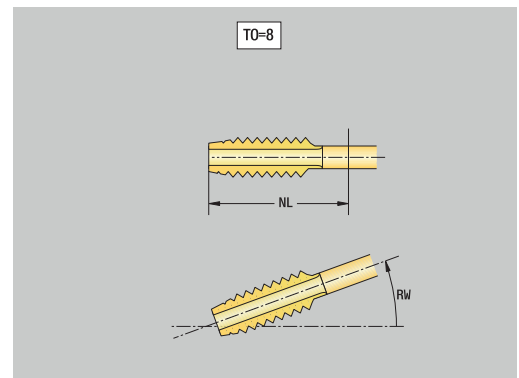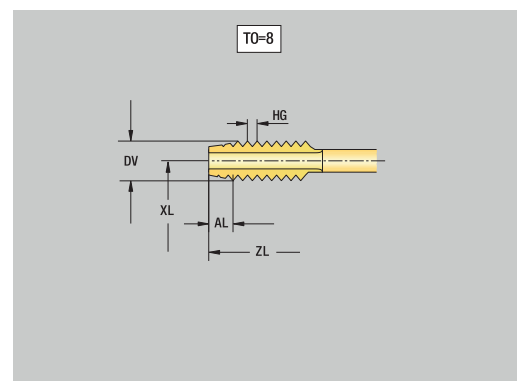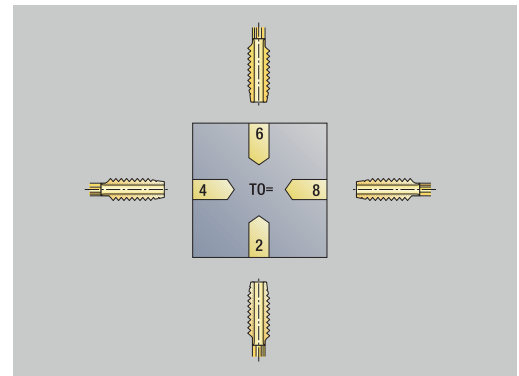
Standard milling tools

New tool
▶ Press the **New tool** soft key

▶ Press the **Milling tool** soft key

The help graphics illustrate the dimensions of the tools.

Special parameters for standard milling tools:

- **DV**: **Milling diameter**
- **AZ**: **No. of teeth**
- **DD**: **Special corr.**
- **SL**: **Cutting length**
- **R2**: **Tool radius 2**
- **DR2**: **Tool radius 2 oversize**
- Further tool parameters:
  Further information: "General tool parameters", Page 588

> **i**
> - For milling operations at **constant cutting speed**, **Cutter diameter DV** is used to calculate the spindle speed
> - For **G193 Feed per tooth**, the **No. of teeth AZ** parameter is evaluated

Thread milling tools

New tool	▶ Press the **New tool** soft key
Special tool	▶ Press the **Special tool** soft key
[milling tool icon]	▶ Press the **Milling tool** soft key
[thread cutter icon]	▶ Press the **Thread cutter** soft key

The help graphics illustrate the dimensions of the tools.

Special parameters for thread milling tools:

- **DV**: **Milling diameter**
- **AZ**: **No. of teeth**
- **FB**: **Cutter width/height**
- **HG**: **Thread pitch**
- **DD**: **Special corr.**
- Further tool parameters:
  Further information: "General tool parameters", Page 588

> ⓘ
> - For milling operations at **constant cutting speed**, **Cutter diameter DV** is used to calculate the spindle speed
> - For **G193 Feed per tooth**, the **No. of teeth AZ** parameter is evaluated

Angle cutters
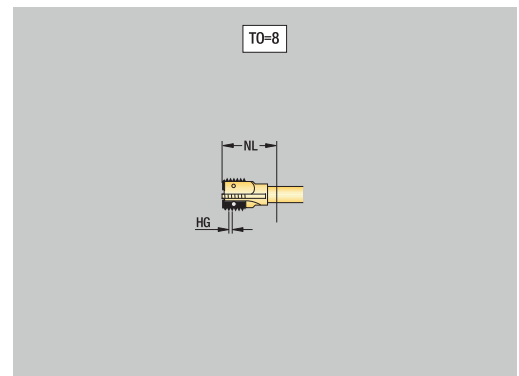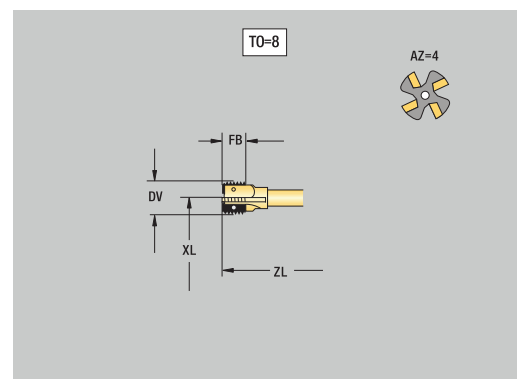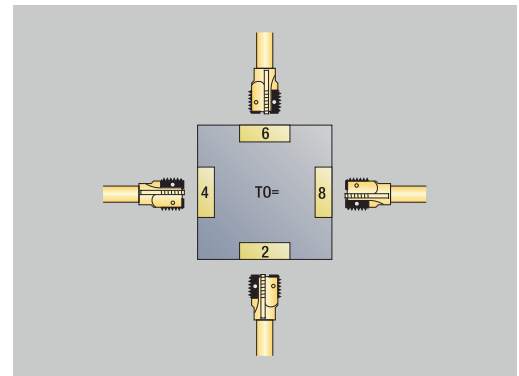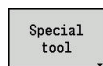
[New tool]	▶ Press the **New tool** soft key
[Special tool]	▶ Press the **Special tool** soft key
[special milling tools icon]	▶ Press the **special milling tools** soft key
[Angle cutter icon]	▶ Press the **Angle cutter** soft key

The help graphics illustrate the dimensions of the tools.

Special parameters for angle cutters:

- **DV**: **Milling diameter**
- **AZ**: **No. of teeth**
- **FB**: **Cutter width/height**
  - **FB** < 0: Large cutter diameter on front
  - **FB** > 0: Large cutter diameter on back
- **FW**: **Cutter angle**
- **DD**: **Special corr.**
- Further tool parameters:
  Further information: "General tool parameters", Page 588

> ⓘ ■ For milling operations at **constant cutting speed**, **Cutter diameter DV** is used to calculate the spindle speed
> ■ For **G193 Feed per tooth**, the **No. of teeth AZ** parameter is evaluated

Burr cutters
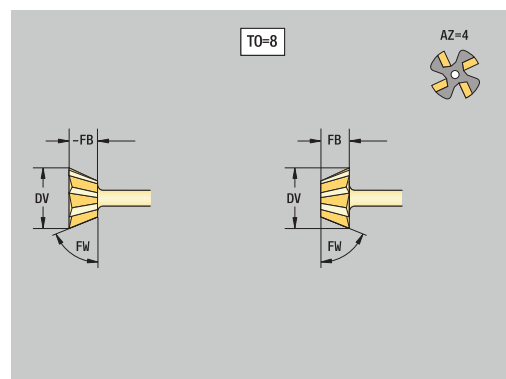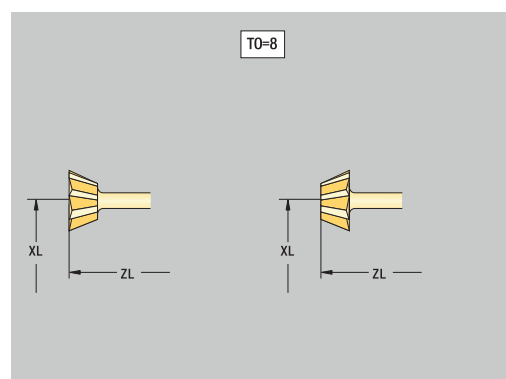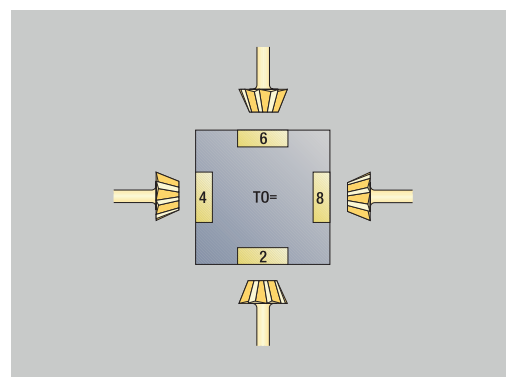
| New tool | ▶ Press the **New tool** soft key |

| Special tool | ▶ Press the **Special tool** soft key |

| | ▶ Press the **special milling tools** soft key |

| | ▶ Press the **Thin millg. cutter** soft key |

The help graphics illustrate the dimensions of the tools.

Special parameters for burr cutters:

- **DV**: **Milling diameter**
- **AZ**: **No. of teeth**
- **SL**: **Cutting length**
- **FW**: **Cutter angle**
- **DD**: **Special corr.**
- Further tool parameters:
  Further information: "General tool parameters", Page 588

> ℹ︎
> - For milling operations at **constant cutting speed**, **Cutter diameter DV** is used to calculate the spindle speed
> - For **G193 Feed per tooth**, the **No. of teeth AZ** parameter is evaluated
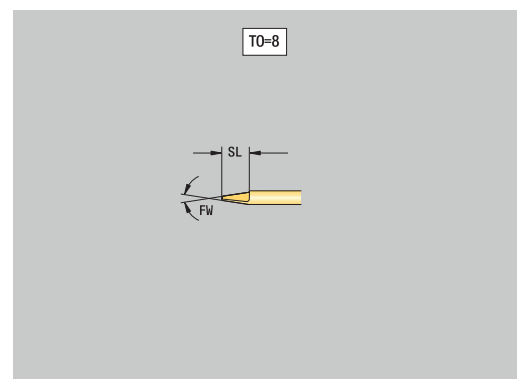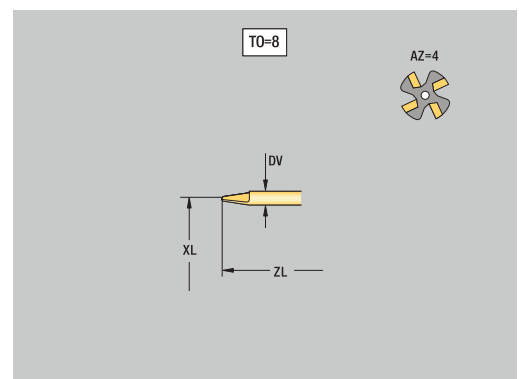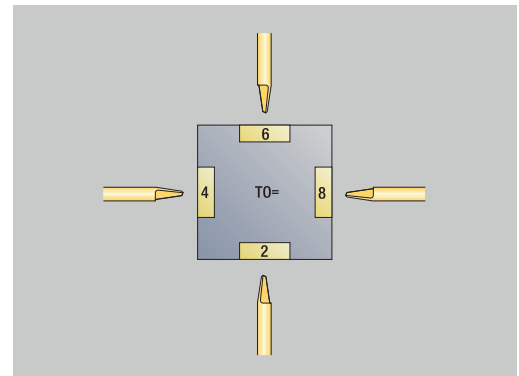
Knurling tool

▶ Press the **New tool** soft key

▶ Press the **Special tool** soft key
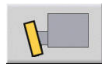
▶ Press the **Knurling tool** soft key

The help graphics illustrate the dimensions of the tools.

Special parameters for knurling tools:

- **SL**: **Cutting length**
- **EW**: **Setting angle**
- **SB**: **Cutting width**
- **DN**: **Tool width**
- **SD**: **Shank diameter**
- Further tool parameters:
  Further information: "General tool parameters", Page 588
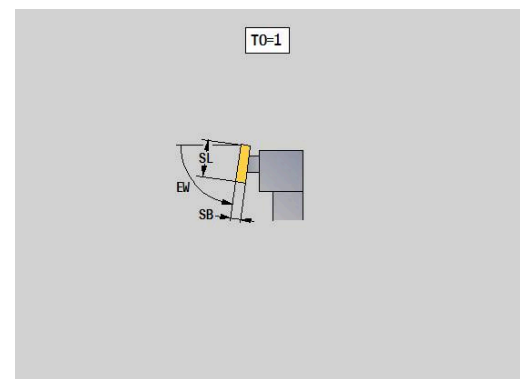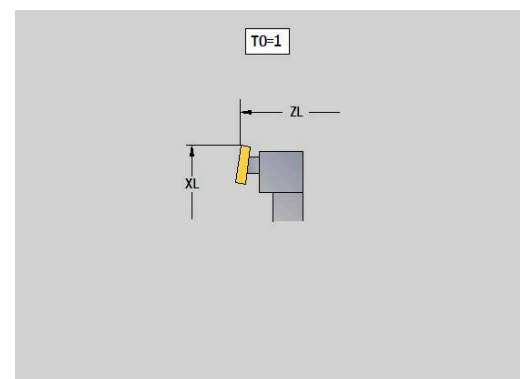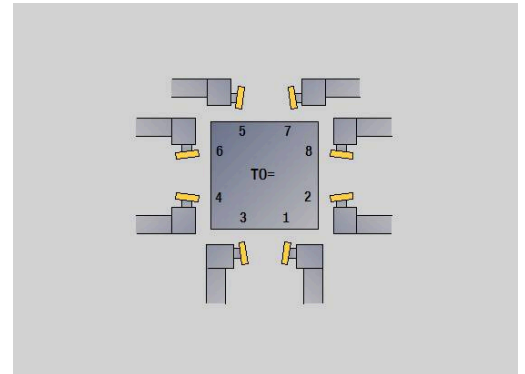
Touch probes

New tool	▶ Press the **New tool** soft key
Special tool	▶ Press the **Special tool** soft key
	▶ Press the **handling systems and touch probes** soft key
	▶ Press the **Touch probe** soft key

The help graphics illustrate the dimensions of the tools.

Special parameters for touch probes:

- **TP**: **Touch probe number** – If a serial number is indicated in the touch probe table, this serial number will be shown in the selection list next to the touch probe type.
- **SD**: **Sphere diameter**
- **CA1**: **Center offset, ref. axis** – Determined based on the **G747** and **G748** calibration cycles
- **CA2**: **Center offset, minor ax.** – Determined based on the **G747** and **G748** calibration cycles
- Further tool parameters:
  Further information: "General tool parameters", Page 588

> 🛈 The **CA1** and **CA2** compensation values can also be edited manually in the tool form.

> ⚙ Refer to your machine manual.
>
> The control must be specially prepared by the machine tool builder for the use of a 3-D touch probe.
>
> HEIDENHAIN gives a warranty for the function of the touch probe cycles only if HEIDENHAIN touch probes are used!
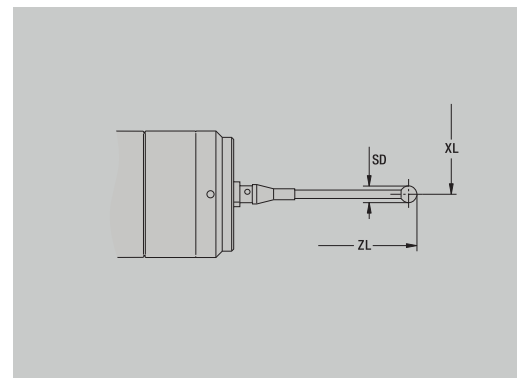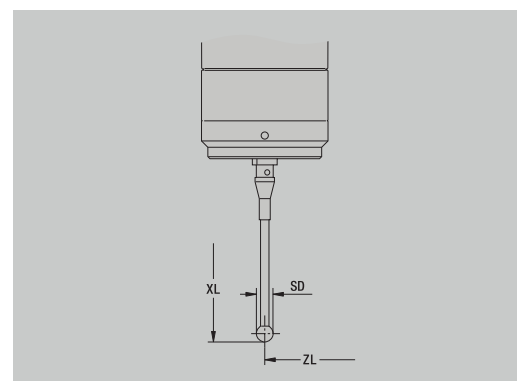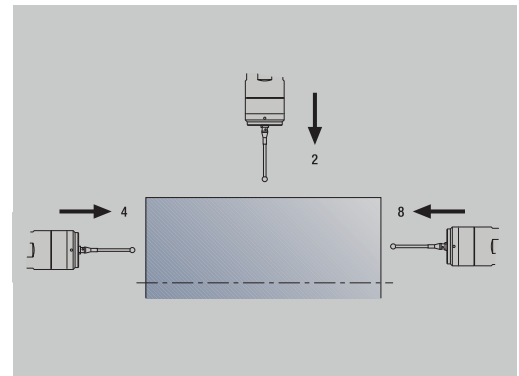
Stopper tools
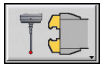
New tool	▶ Press the **New tool** soft key
Special tool	▶ Press the **Special tool** soft key
[icon]	▶ Press the **handling systems and touch probes** soft key
[icon]	▶ Press the **Stopper tool** soft key

The help graphics illustrate the dimensions of the tools.

Special parameters for stopper tools:

- **DD**: **Special corr.**
- Further tool parameters:
  Further information: "General tool parameters", Page 588

Grippers
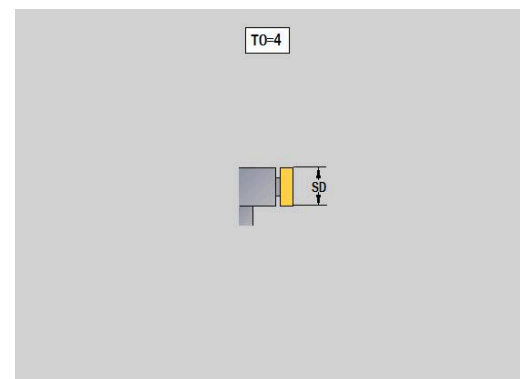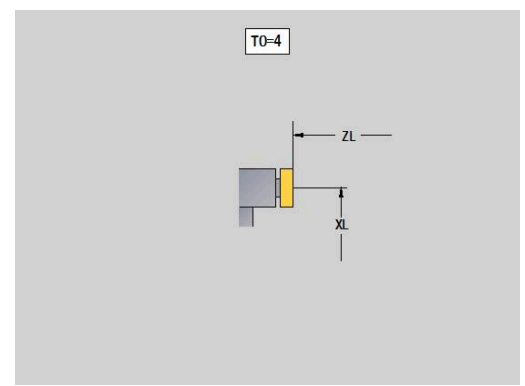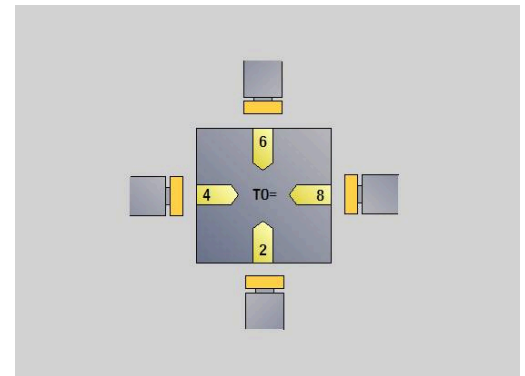
New tool

▶ Press the **New tool** soft key

Special tool

▶ Press the **Special tool** soft key
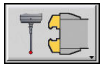
▶ Press the **handling systems and touch probes** soft key
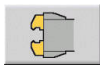
▶ Press the **Gripper** soft key

The help graphics illustrate the dimensions of the tools.

Special parameters for grippers:

- **DD**: **Special corr.**
- Further tool parameters:
  Further information: "General tool parameters", Page 588
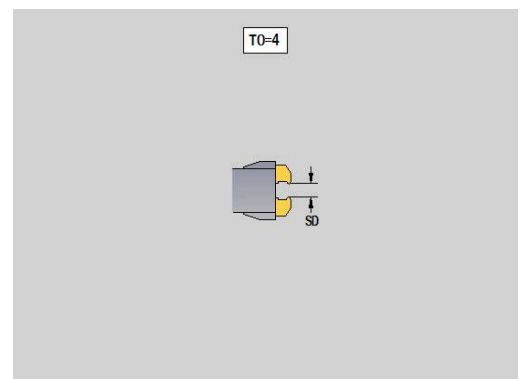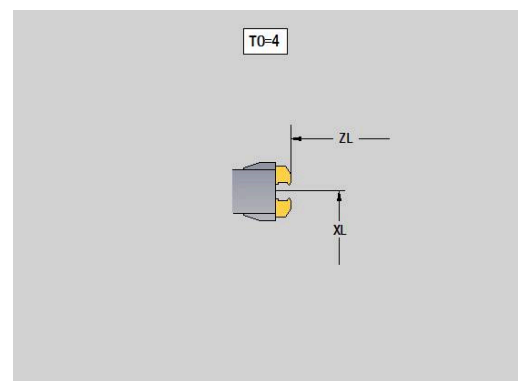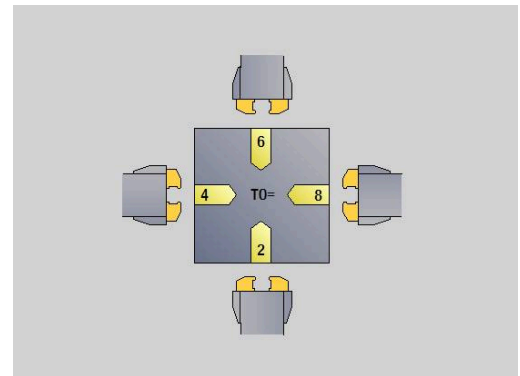
10.4 Technology database

The technology database manages the cutting data according to the machining mode, the workpiece material and the cutting material. The figure at right shows the composition of the database. Each cube represents a data record with cutting data.

In its standard version, the technology database is designed for 9 workpiece-material/cutting-material combinations. With software option **Tools and Technology** (option 10), the database can be expanded to contain up to 62 workpiece-material/cutting-material combinations.

The control ascertains these criteria from the following information:

- **Machining mode**: In cycle programming (**Teach-in** submode), a machining mode is assigned to each cycle, and in **smart.Turn** mode of operation, a machining mode is assigned to each unit

- **Material**: The workpiece material is defined in the **TSF menu** during cycle programming, and in the program header in **smart.Turn** mode of operation

- **Cutting material**: Each tool description contains the cutting material

Using these three criteria, the control accesses a cutting data record (shown yellow in the figure) and generates from it the technology-data proposal.

Explanation of abbreviations used in the figure:

- **Task**: Machining mode
- **WS**: Workpiece material
- **SS**: Cutting material
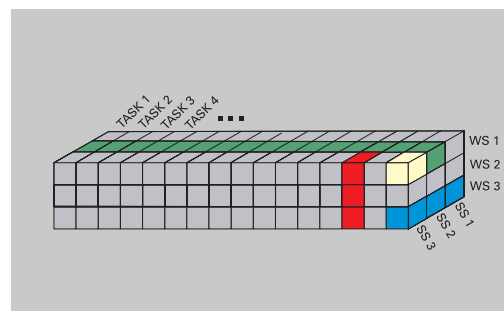
Types of machining

Predrilling	Not used
Roughing	2
Finishing	3
Thread cutting	4
Contour recessing	5
Parting	6
Centering	9
Drilling	8
Countersinking	9
Reaming	Not used
Tapping	11
milling	12
Finish milling	13
Deburring	14
Engraving	15
Recessing	16

Technology editor submode

Technology editor submode can be called from the **Tool editor** and **smart.Turn** operating modes.

Database access to the following combinations is supported:
- Workpiece-material/machining-mode combinations (blue)
- Cutting-material/machining-mode combinations (red)
- Workpiece-material/cutting-material combinations (green)

Editing workpiece-material and cutting-material designations: **Technology editor** submode keeps one list each with workpiece-material designations and cutting-material designations.

You have the following options:
- **Insert** new workpiece/cutting materials
- **Do not change** the workpiece/cutting material designations
- **Delete** existing workpiece/cutting material designations. This also deletes the associated cutting data

Explanation of abbreviations used in the figure:
- **Task**: **Machining mode**
- **WS**: **Material**
- **SS**: **Cutting material**
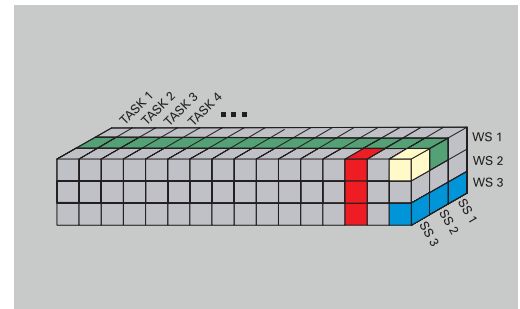
ℹ️ Deleting workpiece-material designations or cutting-material designations also deletes the associated cutting data.
> Following that, the control cannot determine any cutting data for the affected programs or tools.

Editing the cutting data: The cutting data of a workpiece-material/cutting-material combination are called a **data record**.

You have the following options:
- Assign cutting data to a workpiece-material/cutting-material combination to create a new data record
- Delete the cutting data of a workpiece-material/cutting-material combination (a data record)

To call **Technology editor** submode in **Tool editor** operating mode:

[Other tables] ▶ Press the **Other tables** soft key
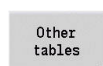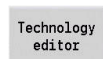
[Technology editor] ▶ Press the **Technology editor** soft key

Editing a workpiece-material or cutting-material list

To edit a workpiece-material list:

▶ Select the **Work materials** menu item

> The editor opens the list with the workpiece-material designations

Adding a workpiece material:

Append work piece mat.

▶ Press the **Append work piece mat.** soft key

▶ Enter the workpiece-material designation (max. 16 characters)

> The sorting number is assigned sequentially

Deleting a workpiece material:

Delete work piece mat.

▶ Press the **Delete work piece mat.** soft key

> After the confirmation request, the control deletes the workpiece material with all associated cutting data

To edit a cutting-material list:

▶ Select the **Cuttg. materl.** menu item

> The editor opens the list with the cutting-material designations

Adding a cutting material:

Append cutting mat.

▶ Press the **Append cutting mat.** soft key

▶ Enter the cutting-material designation (max. 16 characters)

> The sorting number is assigned sequentially

Deleting a cutting material:

Delete cutting mat.

▶ Press the **Delete cutting mat.** soft key

> After the confirmation request, the control deletes the cutting material with all associated cutting data

The sorting number only specifies the sequence within the list.

To change the sorting number:

▶ Select the sorting number

Edit field

▶ Press the **Edit field** soft key

▶ Enter the new number
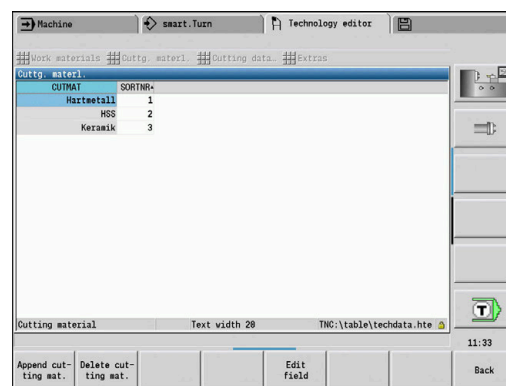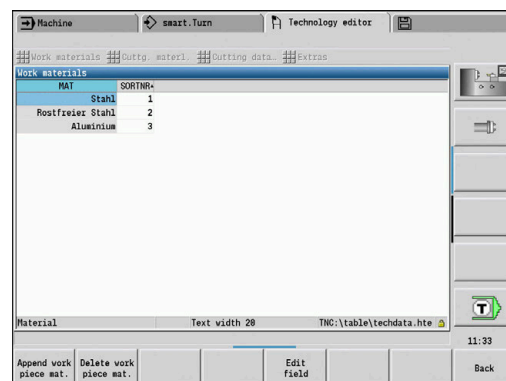
ℹ️ Simply expanding the workpiece-material or cutting-material list does not create cutting data. The data record for cutting data of a new workpiece-material/cutting-material combination is not created until you have requested it through the **New data record** soft key.

Displaying and editing cutting data

To display the cutting data of the machining modes:

> ▶ Select the **Cutting data...** menu item
>
> > The editor opens the dialog for selecting a workpiece material/cutting material combination.
>
> ▶ Select the desired combination
>
> ▶ Press the **OK** soft key
>
> > **Technology editor** submode displays the cutting data.

To display the cutting data of the workpiece materials:

> ▶ Select the **Extras** menu item
>
> ▶ Select the **Work mat. tab...** menu item
>
> > The editor opens the dialog for selecting a machining-mode / cutting-material combination.
>
> ▶ Select the desired combination
>
> ▶ Press the **OK** soft key
>
> > **Technology editor** submode displays the cutting data.

To display the cutting data of the cutting materials:

> ▶ Select the **Extras** menu item
>
> ▶ Select the **Cut. mat. tab...** menu item
>
> > The editor opens the dialog for selecting a workpiece material/cutting material combination.
>
> ▶ Select the desired combination
>
> ▶ Press the **OK** soft key
>
> > **Technology editor** submode displays the cutting data.

ℹ The **value 0** in a data record means that no value is transferred to the unit or cycle dialog.
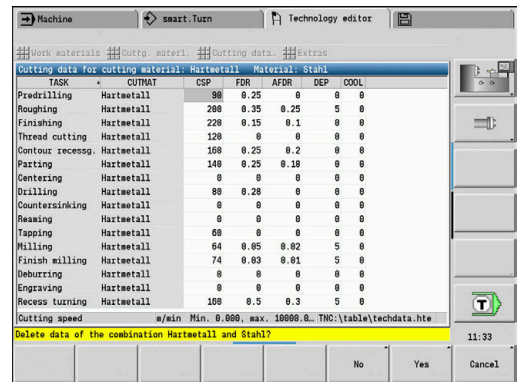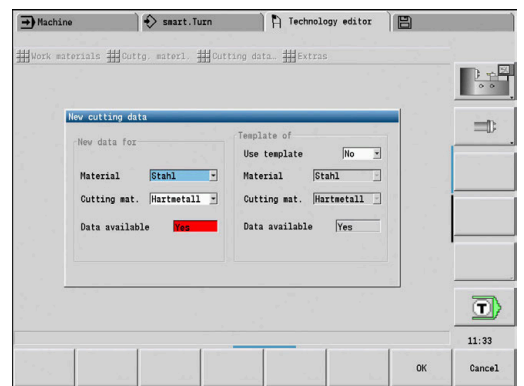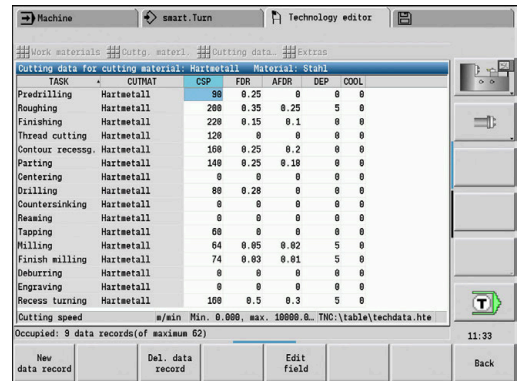
To edit the cutting data:

▶ Call the table with cutting data

| ↓ | ▶ With the arrow keys, select the cutting-data field you want to edit |

| Edit field | ▶ Press the **Edit field** soft key |

▶ Enter a value

| ENT | ▶ Press the **ENT** key |

To add new cutting data:

▶ Select any workpiece-material/cutting material combination
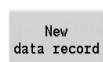
New data record	▶ Press the **New data record** soft key
	> **Technology editor** submode opens the **New cutting data** dialog.
	▶ Select the desired workpiece-material/cutting-material combination
	▶ Decide whether an existing workpiece-material/cutting-material combination should be used as a template. Otherwise, the default value **0** is assigned to all entries
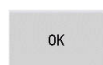
| OK | ▶ Press the **OK** soft key |

To delete a data record with cutting data:

▶ Select the workpiece-material/cutting-material combination (data record) to be deleted
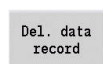
| Del. data record | ▶ Press the **Del. data record** soft key |
| | > **Technology editor** submode prompts you if you really want to delete the data record. |

| Yes | ▶ Press the **YES** soft key |
| | > **Technology editor** submode deletes the data record of the given workpiece-material/cutting-material combination. |

11

Organization Mode
of Operation

11.1 Organization mode of operation

Organization operating mode offers various functions for communication with other systems, data backup, setting of parameters, and diagnosis.

The following functions are available:

- **Login code**: Some parameter settings and functions may only be accessed by authorized personnel. Users need to enter a code number to log in to this mode
- **Parameter settings**: Parameters enable you to adapt the control to your specific requirements. The Machine parameter menu provides functions to view and edit parameters
- **Transfer**: **Transfer** submode is used either for exchange of data with other systems or for data backup. You can input and output programs, parameters, and tool data
- **Diagnosis**: The control provides diagnostic functions for checking the system and for troubleshooting

> **ℹ** The Configuration and Diagnosis functions can only be accessed by authorized commissioning and service personnel.

Software numbers

The following software numbers are displayed on the control's screen after the **Key** soft key has been selected:

- HEIDENHAIN
  - **Control model**: Designation of the control (managed by HEIDENHAIN)
  - **NC-SW**: Number of the NC software (managed by HEIDENHAIN)
  - **NCK**: Number of the NC software (managed by HEIDENHAIN)
- Functional safety
  - **MC-FS**: SKERN software of the MC
  - **CC-FS:** SKERN software of the CC number x
  - **SPLC-SW**: Number of the SPLC program
- PLC
  - **PLC-SW**: Number or name of the PLC software (managed by the machine tool builder)

> **ℹ** The corresponding information will only be displayed if functional safety (FS) is available on your machine.

Login code

Login code (code number)	Possibilities
	- Edit the selected machine parameters **Transfer** submode: - Transmit or receive programs - Create service files
123	- Edit all machine parameters **Transfer** submode: - Parameter backup - Back up and restore tools
sik	- Options dialog - Dialog for activating software options in the SIK (system identification key)
Service code	- Edit configuration data - Diagnostic functions - Restore parameters

11.2 Parameters

Parameter editor

The parameter values are entered in the configuration editor.

Each parameter object has a name, e.g. **CfgDisplayLanguage** (101300) that gives information about the parameters it contains. Each object has a **key** for unique identification.

The icon at the beginning of each line in the parameter tree shows additional information about this line. The icons have the following meanings.

Icon	Meaning
	Branch exists but is closed
	Branch is open
	Empty object, cannot be opened
	Initialized machine parameter
	Uninitialized (optional) machine parameter
	Can be read but not edited
	Can neither be read nor edited

Machine parameters (user parameters)

Machine parameters are used to adapt the behavior of the control.

Machine parameters that are preset for the usual "daily operations" are organized as user parameters.

> Refer to your machine manual.
>
> Your machine tool builder can make additional machine parameters available as user parameters.
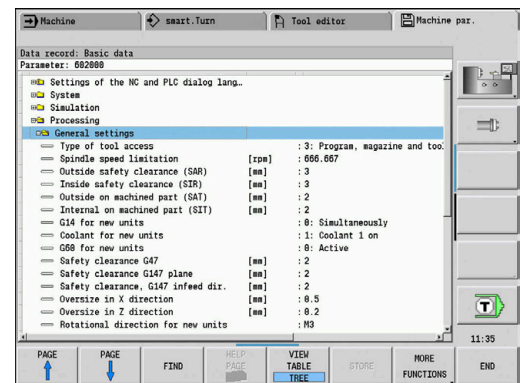
Editing machine parameters

To edit the machine parameters:

▶ Press the **Key** soft key

▶ Enter the code number 123

▶ Alternative: Press the **USER PARAMETER** soft key

Displaying help texts

To show a help text:

▶ Position the cursor on the parameter

▶ Press the **Info** key

> The parameter editor opens a window with information on this parameter.

▶ Press the **Info** key again

> The parameter editor closes the window with information on this parameter.
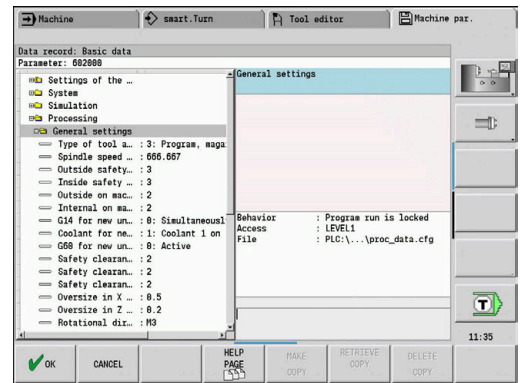
Finding parameters

To find parameters:

▶ Press the **FIND** soft key

▶ Enter the search criteria

▶ Press the **FIND** soft key again
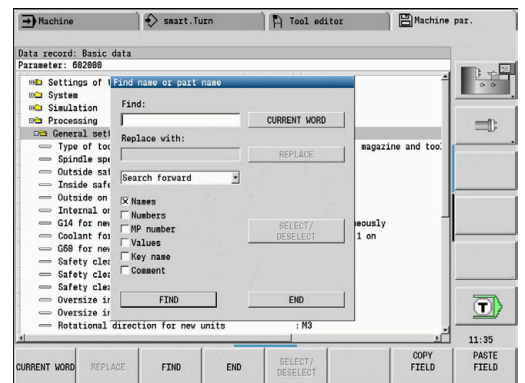
Exiting the parameter editor

To exit the parameter editor:

▶ Press the **END** soft key

List of machine parameters

Language setting

Definition of the NC and PLC conversational languages (no. 101300)

　　NC conversational language (no. 101301)

　　　　ENGLISH

　　　　GERMAN

　　　　CZECH

　　　　FRENCH

　　　　ITALIAN

　　　　SPANISH

　　　　PORTUGUESE

　　　　SWEDISH

　　　　DANISH

　　　　FINNISH

　　　　DUTCH

　　　　POLISH

　　　　HUNGARIAN

　　　　RUSSIAN

　　　　CHINESE

　　　　CHINESE_TRAD

　　　　SLOVENIAN

　　　　KOREAN

　　　　NORWEGIAN

　　　　ROMANIAN

　　　　SLOVAK

　　　　TURKISH

　　Load the language of the NC control (no. 101305)

　　　　FALSE: When starting the control, the language of the HEROS operating system is used

　　　　TRUE: When starting the control, the language from the machine parameters is used

　　PLC conversational language (no. 101302)

　　　　See NC conversational language

　　PLC error message language (no. 101303)

　　　　See NC conversational language

　　Help language (no. 101304)

　　　　See NC conversational language

General settings

System

　　Definition of the unit of measure valid for the display (101100)

　　　　Unit of measure for display and user interface (101101)

　　　　　　metric: Use metric system

　　　　　　inch: Use inch system

General settings

System

General display settings (604800)

Axis display (604803)

Default

REFIST: Actual value

REF.NOML.: Nominal value

LAG: Servo lag

DIST: Distance to go

File preview during program selection(604804)

TRUE: File preview during program selection is displayed

FALSE: No file preview during program selection

Do not display limit switch warnings (604805)

TRUE: If an axis is positioned on the limit switch position, no limit switch warning will be displayed

FALSE: A limit switch warning will be displayed

System

Settings for automatic operation (601800)

Tool life management (601801)

On: Tool life management is active

Off: Tool life management is inactive

Program run with the most recently selected cycle (601809)

On: When Program Run is selected, the most recently selected cycle remains active.

Off: When Program Run is selected, the first cycle will be active.

End mid-program startup after start block (601810)

TRUE: In mid-program startup, program execution begins with the following NC block

FALSE: In mid-program startup, program execution begins with the selected NC block

Special dimensioning of button tools (601812)

On: When you enter a tool angle of 90 degrees, the tool tip of a button tool that has no neutral tool orientation (TO not equal to 2, 4, 6, 8), will be dimensioned like a neutral tool

Off: When you enter a tool angle of 90 degrees, the tool tip of a button tool that has no neutral tool orientation (TO not equal to 2, 4, 6, 8), will be dimensioned with respect to the point on the tool tip that corresponds to the orientation

Automatic program selection (601814)

On: When you activate Program Run mode of operation, the program that was last active will be selected

Off: When you activate Program Run mode of operation, the program that was last active will not be selected

General settings

System

Tool dimensioning (604600)

Measuring feed rate (604602)

[mm/min]: Feed rate for the approach of the touch probe

Measuring range (604603)

[mm]: The touch probe must be deflected within the measuring range. Otherwise, an error message will be issued

System

Settings for Machine mode of operation (604900)

Save cycle without simulation (604903)

TRUE: The cycle can be saved without previous simulation or execution

FALSE: The cycle can only be saved after previous simulation or execution

Executed tool change with NC start (604904)

TRUE: A tool change using the TSF dialog will be executed with the Start cycle

FALSE: No tool change will not be executed with the Start cycle

Separate dialogs for tool change, spindle speed and feed rate (TSF) (604906)

TRUE: Input of tool change, spindle speed, and feed rate data in separate dialogs

FALSE: Input of all cutting data in the TSF dialog

System

Settings for load monitoring (124700)

Activate load monitoring (124701)

TRUE: Load monitoring is active

FALSE: Load monitoring is inactive

Factor for utilization limit 1 (124702)

[%]: This value, multiplied by the reference value determined during reference machining, is the load utilization limit 1

Factor for utilization limit 2 (124703)

[%]: This value, multiplied by the reference value determined during reference machining, is the utilization limit 2

Factor for the limit of the utilization sum (124704)

[%]: This value, multiplied by the reference value determined during reference machining, is the limit value of the utilization sum

Channel settings

Channels

CH_NC1

Configuration of machining cycles (no. 201000)

Do not show the residual material warning (no. 201010)

Never: The warning is always displayed

NCOnly: The warning is suppressed during program run

Always: The warning is never displayed

Settings for the simulation

Simulation

General settings (no. 114800)

Restart with M99 (no. 114801)

On: Simulation restarts at beginning of program

Off: Simulation stops

Path delay (no. 114802)

[s]: Waiting time after each path has been graphically simulated. This influences the simulation speed.

Software limit switches active (no. 114803)

On: Software limit switches also active during simulation

Off: Software limit switches not active during simulation

Deactivate program graphics (no. 114805)

TRUE: Permanently deactivate the program graphics menu item in Program Run submode

FALSE: Deactivate the program graphics menu item in Program Run submode temporarily (i.e., while the currently selected NC program is running)

Simulation

Processing times for NC functions (general) (115000)

Time allowance for tool change (115001)

[s]: These times are used as non-machining times for the time calculation function

Time allowance for gear shifting (115002)

[s]: These times are used as non-machining times for the time calculation function

General time allowance for M functions (115003)

[s]: These times are used as non-machining times for the time calculation function

Settings for the simulation

Simulation

    Processing times for M functions (115100): Individual time allowances for a maximum of 20 M functions

        T01 (115100)

            Number of the M function

            Time allowance for the M function

                [s]: Time calculation adds this time to the general time allowance for M functions

        TXX (115100)

            See T01

Simulation

    Definition of the (default) window size (115200): The simulation adapts the window size to the workpiece blank. If no workpiece blank has been programmed, the simulation will use the default window size

        Datum position in X (115201)

            [mm]: Distance of the coordinate origin from the lower edge of the window

        Datum position in Z (115202)

            [mm]: Distance of the coordinate origin from the left edge of the window

        Delta X (115203)

            [mm]: Vertical extension of the graphic window

        Delta Z (115204)

            [mm]: Horizontal extension of the graphic window

Simulation

    Definition of the (default) workpiece blank size (115300): If no workpiece blank has been programmed in DIN PLUS, the simulation will use the default workpiece blank

        Outside diameter (115301)

            [mm]

        Length of the workpiece blank (115302)

            [mm]

        Right edge of the workpiece blank (115303)

            [mm]

        Inside diameter (115304)

            [mm]

Settings for fixed cycles and units

Processing

General settings (no. 602000)

Type of tool access (no. 602001)

0: First from NC program, then from tool table

1: Only from the NC program

2: First from the NC program, then from the magazine

3: First from the NC program, then from the magazine, then from the tool table

External safety clearance (SAR) (no. 602005)

[mm]: External safety clearance on workpiece blank

Internal safety clearance (SIR) (no. 602006)

[mm]: Internal safety clearance on workpiece blank

External on machined part (SAT) (no. 602007)

[mm]: External safety clearance on machined part

Internal on machined part (SIT) (no. 602008)

[mm]: Internal safety clearance on machined part

G14 for new units (no. 602009): Default value for tool change position G14

No axis

0: Simultaneously

1: First X, then Z

2: First Z, then X

3: Only X

4: Only Z

5: Only Y

6: Simultaneous with Y

Coolant for new units (no. 602010): Default value for coolant (CLT)

0: Without

1: Circuit 1 on

2: Circuit 2 on

G60 for new units (no. 602011): Default value for protection zone G60

0: Active

1: Inactive

Safety clearance G47 (no. 602012)

[mm]: Default value for safety clearance G47

Safety clearance G147 plane (no. 602013)

[mm]: Default value for safety clearance (SCK)

Safety clearance G147 in infeed direction (no. 602014)

[mm]: Default value for safety clearance (SCI)

Settings for fixed cycles and units

Oversize in X direction (no. 602015)
[mm]: Default value for oversize (X) I

Oversize in Z direction (no. 602016)
[mm]: Default value for oversize (Z) K

Rotational direction for new units (no. 602017): Default value for direction of rotation (MD)
M3
M4

Datum shift (no. 602022)
On: The AWG generates a datum shift
Off: The AWG does not generate a datum shift

Front edge of chuck on main spindle (no. 602018)
[mm]: Z position of the front edge of the chuck for calculating the workpiece datum

Front chuck edge on counter spindle (no. 602019)
[mm]: Z position of the front edge of the chuck for calculating the workpiece datum

Jaw width on main spindle (no. 602020)
[mm]: Jaw width in Z direction for calculating the workpiece datum

Jaw width on counter spindle (no. 602021)
[mm]: Jaw width in Z direction for calculating the workpiece datum

Conversion of the ICP contours (no. 602023)
0: Read out calculated parameters
1: Read out programmed parameters

Generation of contour groups (no. 602024)
OFF: The AWG does not generate contour groups
ON: The AWG generates two contour groups for full-surface machining with dual spindles

Generation of structure program (no. 602025)
OFF: The AWG does not generate a structure program
ON: The AWG generates a structure program

Deletion of contour group on the counter spindle (no. 602026)
OFF: The contour group on the counter spindle is not deleted
ON: The AWG deletes the contour group on the counter spindle

Oversize for tool holder G891 (no. 602027)
[mm]

Settings for fixed cycles and units

Processing

Global parameters for finished parts (601900)

Max. inward copying angle (EKW) (601903)

[°]: Limit angle for the distinction between turning and recessing

Settings for fixed cycles and units

Processing

Centric predrilling (602100)

First drill limit diameter [UBD1] (602101)

[mm]: Limit diameter for the first predrilling pass

Second drill limit diameter [UBD2] (602102)

[mm]: Limit diameter for the second predrilling pass

Point angle tolerance [SWT] (602103)

[°]: Permissible point angle deviations for oblique drilling limit elements

Drilling oversize – diameter [BAX] (602104)

[mm]: Machining oversize for drilling diameter in X-direction (radius dimension)

Drilling oversize – depth [BAZ] (602105)

[mm]: Machining oversize for drilling depth in Z-direction

Approach to predrilling [ANB] (602106): Approach strategy

1: XZ – Simultaneously

2: XZ – Sequentially

3: ZX – Sequentially

Departure for tool change [ABW] (602106): Departing strategy

1: XZ – Simultaneously

2: XZ – Sequentially

3: ZX – Sequentially

Safety clearance to workpiece blank [SAB] (602108)

[mm]: Safety clearance to workpiece blank

Safety clearance, inside [SIB] (602109)

[mm]: Retraction clearance for deep drilling (B in G74)

Drilling depth ratio [BTV] (602110)

Ratio for checking the predrilling passes (BTV <= BT/dmax)

Drilling depth factor [BTF] (602111)

Factor for the calculation of the first drilling depth for deep drilling (bt1 = BTF*db)

Drilling depth reduction [BTR] (602112)

[mm]: Reduction for deep drilling (bt2 = bt1 – IBTR)

Overhang length – predrilling [ULB] (602113)

[mm]: Default value for spot drilling/through hole A

Settings for fixed cycles and units

Processing

Roughing (no. 602200)

Tool angle – outside/longitudinal [RALEW] (no. 602201)

[°]: Tool angle for roughing tool

Point angle – outside/longitudinal [RALSW] (no. 602202)

[°]: Point angle for roughing tool

Tool angle – outside/transverse [RAPEW] (no. 602203)

[°]: Tool angle for roughing tool

Point angle – outside/transverse [RAPSW] (no. 602204)

[°]: Point angle for roughing tool

Tool angle – inside/longitudinal [RILEW] (no. 602205)

[°]: Tool angle for roughing tool

Point angle – inside/longitudinal [RILSW] (no. 602206)

[°]: Point angle for roughing tool

Tool angle – inside/transverse [RIPEW] (no. 602207)

[°]: Tool angle for roughing tool

Point angle – inside/transverse [RIPSW] (no. 602208)

[°]: Point angle for roughing tool

Machining outside/longitudinal [RAL] (no. 602209): Roughing strategy

Complete: Full-surface roughing cycle including plunge-cutting
Standard: Standard roughing cycle without plunge-cutting

Machining inside/longitudinal [RIL] (no. 602210): Roughing strategy

Complete: Full-surface roughing cycle including plunge-cutting
Standard: Standard roughing cycle without plunge-cutting

Machining outside/transverse [RAP] (no. 602211): Roughing strategy

Complete: Full-surface roughing cycle including plunge-cutting
Standard: Standard roughing cycle without plunge-cutting

Machining inside/transverse [RIP] (no. 602212): Roughing strategy

Complete: Full-surface roughing cycle including plunge-cutting
Standard: Standard roughing cycle without plunge-cutting

Secondary angle tolerance [RNWT] (no. 602213)

[°]: Tolerance range for secondary cutting edge

Relief cutting angle [RFW] (no. 602214)

[°]: Minimum difference between contour and secondary cutting edge

Type of oversize [RAA] (no. 602215)

Settings for fixed cycles and units

16: Longitudinal and transverse oversizes differ – No single oversizes

32: Equidistant oversize – No single oversizes

144: Longitudinal and transverse oversizes differ – With single oversizes

160: Equidistant oversize – With single oversizes

Equidistant or longitudinal [RLA] (no. 602216)

[mm]: Equidistant oversize or longitudinal oversize

Transverse oversize [RPA] (no. 602217)

[mm]: Transverse oversize

Approach / outside roughing [ANRA] (no. 602218): Approach strategy

1: XZ – Simultaneous

2: XZ – Sequential

3: ZX – Sequential

Approach / inside roughing [ANRI] (no. 602219): Approach strategy

1: XZ – Simultaneous

2: XZ – Sequential

3: ZX – Sequential

Departure / outside roughing [ABRA] (no. 602220): Departure strategy

1: XZ – Simultaneous

2: XZ – Sequential

3: ZX – Sequential

Departure / inside roughing [ABRI] (no. 602221): Departure strategy

1: XZ – Simultaneous

2: XZ – Sequential

3: ZX – Sequential

Transverse / longitudinal ratio – outside [PLVA] (no. 602222)

Ratio for determination of longitudinal or transverse machining

Transverse / longitudinal ratio – inside [PLVI] (no. 602223)

Ratio for determination of longitudinal or transverse machining

Minimum transverse length [RMPL] (no. 602224)

[mm]: Radius value for determination of the machining operation

Transverse angle deviation [PWA] (no. 602225)

[°]: Tolerance range in which the first element is declared a transverse element

Overhang length – outside [ULA] (no. 602226)

[mm]: Length by which outside roughing is performed beyond the target position

Overhang length – inside [ULI] (no. 602227)

[mm]: Length by which inside roughing is performed beyond the target position

Settings for fixed cycles and units

Lift-off distance – outside [RAHL] (no. 602228)

[mm]: Lift-off distance for smoothing variants H = 1 and H = 2

Lift-off distance – inside [RIHL] (no. 602229)

[mm]: Lift-off distance for smoothing variants H = 1 and H = 2

Cutting depth reduction factor [SRF] (no. 602230)

Factor that reduces the infeed (cutting depth) for tools that are not used in the main machining direction

Settings for fixed cycles and units

Processing

Finishing (no. 602300)

Tool angle - outside/longitudinal [FALEW] (no. 602301)

[°]: Tool angle for finishing tool

Point angle - outside/longitudinal [FALSW] (no. 602302)

[°]: Point angle for finishing tool

Tool angle - outside/transverse [FAPEW] (no. 602303)

[°]: Tool angle for finishing tool

Point angle - outside/transverse [FAPSW] (no. 602304)

[°]: Point angle for finishing tool

Tool angle - inside/longitudinal [FILEW] (no. 602305)

[°]: Tool angle for finishing tool

Point angle - inside/longitudinal [FILSW] (no. 602306)

[°]: Point angle for finishing tool

Tool angle - inside/transverse [FIPEW] (no. 602307)

[°]: Tool angle for finishing tool

Point angle - inside/transverse [FIPSW] (no. 602308)

[°]: Point angle for finishing tool

Machining outside/longitudinal [FAL] (no. 602309): Finishing strategy

Complete: Full-surface finishing with optimum tool
Standard: Standard finishing; relief turns and undercuts machined with a suitable tool

Machining inside/longitudinal [FIL] (no. 602310): Finishing strategy

Complete: Full-surface finishing with optimum tool
Standard: Standard finishing; relief turns and undercuts machined with a suitable tool

Machining outside/transverse [FAP] (no. 602311): Finishing strategy

Complete: Full-surface finishing with optimum tool
Standard: Standard finishing; relief turns and undercuts machined with a suitable tool

Machining inside/transverse [FIP] (no. 602312): Finishing strategy

Complete: Full-surface finishing with optimum tool
Standard: Standard finishing; relief turns and undercuts machined with a suitable tool

Secondary angle tolerance [FNWT] (no. 602313)

[°]: Tolerance range for secondary cutting edge

Relief cutting angle [FFW] (no. 602314)

[°]: Minimum difference between contour and secondary cutting edge

Approach / outside finishing [ANFA] (no. 602315): Approach strategy

Settings for fixed cycles and units

        1: XZ – Simultaneous

        2: XZ – Sequential

        3: ZX – Sequential

Approach / inside finishing [ANFI] (no. 602316): Approach strategy

        1: XZ – Simultaneous

        2: XZ – Sequential

        3: ZX – Sequential

Departure / outside finishing [ABFA] (no. 602317): Departure strategy

        1: XZ – Simultaneous

        2: XZ – Sequential

        3: ZX – Sequential

Departure / inside finishing [ABFI] (no. 602318): Departure strategy

        1: XZ – Simultaneous

        2: XZ – Sequential

        3: ZX – Sequential

Min. finishing transverse length [FMPL] (no. 602319)

        [mm]: dimension for determining the type of machining

Max. cutting depth for finishing [FMST] (no. 602320)

        [mm]: Permissible infeed depth for non-machined undercuts

No. of revs for chamfer/rounding [FMUR] (no. 602321)

        Minimum number of revolutions; feed rate is reduced automatically

Check of cutting-edge length (no. 602322)

        On: Checks whether the usable cutting edge length is sufficient for finishing

        Off: No check whether the usable cutting edge length is sufficient for finishing

Settings for fixed cycles and units

Processing

  Recessing (no. 602400)

    Approach / External recessing [ANESA] (no. 602401): Approach strategy

      1: XZ – Simultaneous
      2: XZ – Sequential
      3: ZX – Sequential

    Approach / Internal recessing [ANESI] (no. 602402): Approach strategy

      1: XZ – Simultaneous
      2: XZ – Sequential
      3: ZX – Sequential

    Departure / External recessing [ABESA] (no. 602403): Departure strategy

      1: XZ – Simultaneous
      2: XZ – Sequential
      3: ZX – Sequential

    Departure / Internal recessing [ABESI] (no. 602404): Departure strategy

      1: XZ – Simultaneous
      2: XZ – Sequential
      3: ZX – Sequential

    Approach / External contour recessing [ANKSA] (no. 602405): Approach strategy

      1: XZ – Simultaneous
      2: XZ – Sequential
      3: ZX – Sequential

    Approach / Internal contour recessing [ANKSI] (no. 602406): Approach strategy

      1: XZ – Simultaneous
      2: XZ – Sequential
      3: ZX – Sequential

    Departure / External contour recessing [ABKSA] (no. 602407): Departure strategy

      1: XZ – Simultaneous
      2: XZ – Sequential
      3: ZX – Sequential

    Departure / Internal contour recessing [ABKSI] (no. 602408): Departure strategy

      1: XZ – Simultaneous
      2: XZ – Sequential
      3: ZX – Sequential

    Recessing width divisor [SBD] (no. 602409)

      Value for selecting the tool for contour recessing with linear elements on the recessing floor

    Type of oversize [KSAA] (no. 602410)

      16: Longitudinal and transverse oversizes differ – No single oversizes

Settings for fixed cycles and units

32: Equidistant oversize – No single oversizes

144: Longitudinal and transverse oversizes differ – With single oversizes

160: Equidistant oversize – With single oversizes

Equidistant or longitudinal [KSLA] (no. 602411)

[mm]: Equidistant oversize or longitudinal oversize

Transverse oversize [KSPA] (no. 602412)

[mm]: Transverse oversize

Recessing width factor [SBF] (no. 602413)

Factor for determining the maximum tool offset

Recessing/Finishing (no. 602414): Sequence of finishing cuts

1: Divide the paraxial floor elements in half (previous behavior)

2: Move through with lift-off

Processing

Thread cutting (602500)

Approach/thread cutting, outside [ANGA] (602501): Approach strategy

1: XZ – Simultaneously

2: XZ – Sequentially

3: ZX – Sequentially

Approach/thread cutting, inside [ANGI] (602502): Approach strategy

1: XZ – Simultaneously

2: XZ – Sequentially

3: ZX – Sequentially

Departing/thread cutting, outside [ABBS] (602503): Departing strategy

1: XZ – Simultaneously

2: XZ – Sequentially

3: ZX – Sequentially

Departing/thread cutting, inside [ABGI] (602504): Departing strategy

1: XZ – Simultaneously

2: XZ – Sequentially

3: ZX – Sequentially

Thread run-in length [GAL] (602505)

[mm]: Default value for the run-in length B

Thread run-out length [GUL] (602506)

[mm]: Default value for the run-out length (ending length) P

Settings for fixed cycles and units

Processing

    Measuring (602600)

        Measuring loop counter [MC] (602602)

            Specifies the measuring intervals

        Measuring departing length in Z [MLZ] (602603)

            [mm]: Departing length in the Z direction

        Measuring departing length in X [MLX] (602604)

            [mm]: Departing length in the X direction

        Measuring allowance [MA] (602605)

            [mm]: Allowance on the element to be measured

        Measuring cut length [MSL] (602606)

            [mm]: Measuring cut length

Settings for fixed cycles and units

Processing

    Drilling (602700)

        Approach/front face – drilling [ANBS] (602701): Approach strategy

           1: XZ – Simultaneously

           2: XZ – Sequentially

           3: ZX – Sequentially

        Approach/lateral surface – drilling [ANBM] (602702): Approach strategy

           1: XZ – Simultaneously

           2: XZ – Sequentially

           3: ZX – Sequentially

        Departing/front face – drilling [ABBS] (602703): Departing strategy

           1: XZ – Simultaneously

           2: XZ – Sequentially

           3: ZX – Sequentially

        Departing/lateral surface – drilling [ABBM] (602704): Departing strategy

           1: XZ – Simultaneously

           2: XZ – Sequentially

           3: ZX – Sequentially

        Inside safety clearance [SIBC] (602705)

           [mm]: Retraction clearance for deep drilling B

        Driven drilling tool [SBC] 602706)

           [mm]: Safety clearance for driven tools

        Non-driven drilling tool [SBCF] (602707)

           [mm]: Safety clearance for tools that are not driven

        Driven tap [SGC] (602708)

           [mm]: Safety clearance for driven tools

        Non-driven tap [SGCF] (602709)

           [mm]: Safety clearance for tools that are not driven

        Drilling depth factor [BTCF] (602710)

           Factor for the calculation of the first drilling depth for deep drilling

        Drilling depth reduction [BTRC] (602711)

           [mm]: Reduction for deep drilling

        Diameter tolerance for drills [BDT] (602712)

           [mm]: Used for the selection of the tool for drilling

Settings for fixed cycles and units

Processing

    Milling (no. 602800)

        Approach/face – milling [ANMS] (no. 602801): Approach strategy

            1: XZ – Simultaneous

            2: XZ – Sequential

            3: ZX – Sequential

        Approach/lateral surface – milling [ANMM] (no. 602802): Approach strategy

            1: XZ – Simultaneous

            2: XZ – Sequential

            3: ZX – Sequential

        Departure/face – milling [ABMS] (no. 602803): Departure strategy

            1: XZ – Simultaneous

            2: XZ – Sequential

            3: ZX – Sequential

        Departure/lateral surface – milling [ABMM] (no. 602804): Departure strategy

            1: XZ – Simultaneous

            2: XZ – Sequential

            3: ZX – Sequential

        Safety clearance in infeed dir. [SMZ] (no. 602805)

            [mm]: Distance between starting position and top edge of object to be milled

        Safety clearance, milling dir. [SME] (no. 602806)

            [mm]: Distance between milling contour and edge of milling cutter

        Oversize in milling direction [MEA] (no. 602807)

            [mm]: Oversize

        Oversize in infeed direction [MZA] (no. 602808)

            [mm]: Oversize

        Overlap factor (trochoidal milling) (no. 602809)

            Proportional distance of overlapping milling paths

        Retraction feed rate (trochoidal milling) (no. 602819)

            Retraction feed rate in trochoidal milling

        Retraction distance (trochoidal milling) (no. 602811)

            [mm]: Liftoff distance during retraction in trochoidal milling

        Plunge angle (no. 602812)

            [°]: Angle to be used for plunging on a helix or for reciprocating

        Helix diameter factor (no. 602813)

            Proportional factor by which the plunging helix is greater than the cutter diameter

Settings for fixed cycles and units

Infeed rate factor (trochoidal milling) (no. 602814)
Proportional factor by which the feed rate should deviate from the nominal value

Feed rate factor (trochoidal milling) (no. 602815)
Proportional factor by which the feed rate should deviate from the nominal value

Overlap factor (area milling) (no. 602816)
Proportional distance of overlapping milling paths

Overrun factor (area milling) (no. 602817)
Proportional factor by which the cutter mills beyond the surface

Infeed rate factor (area milling) (no. 602818)
Proportional factor by which the feed rate should deviate from the nominal value

Processing
ExpertPrograms
Expert programs (606800)
Parameter list
Key in parameter list

Parameter lists for expert programs (606900)
Name of the expert program
Name of the expert program without path

Parameter
Parameter value

Processing
ProgramUnits
Define units (607000)
Template list

Descriptions of the most important machining parameters (Processing)

> **ⓘ** Machining parameters are used by the working plan generation (**AWG**) (**TURN PLUS** operating mode) and various machining cycles.

General settings

Global technology parameters – Safety clearances

Global safety clearances

Parameter	Meaning
■ Outside on workpiece blank **SAR** ■ Inside on workpiece blank **SIR**	**TURN PLUS** takes **SAR** and **SIR** into account: ■ for all roughing operations ■ for centric predrilling
■ Outside on machined part **SAT** ■ Inside on machined part **SIT**	**TURN PLUS** considers **SAT** and **SIT** on premachined workpieces for: ■ Finish-machining ■ Recess turning ■ Contour recessing (contour cutting) ■ Recessing ■ Thread cutting ■ Measuring
Protection zone **G60** for new units	Default setting for the protection zone (start unit: **G60** parameter): ■ 0: Active ■ 1: Inactive
Global safety clearance **G47**	Default setting for the global safety clearance (start unit: **G47** parameter)
Global safety clearance **G147** in the plane	Default setting for the global safety clearance in the plane (start unit: **SCK** parameter)
Global safety clearance **G147** in infeed direction	Default setting for the global safety clearance in infeed direction (start unit: **SCI** parameter)
Global oversize in X direction	Default setting for the global safety clearance in X direction (start unit: **I** parameter)
Global oversize in Z direction	Default setting for the global safety clearance in Z direction (start unit: **K** parameter)
Front edge of chuck on main spindle	Z position of the front edge of the chuck for calculating the workpiece datum (**AWG** submode)
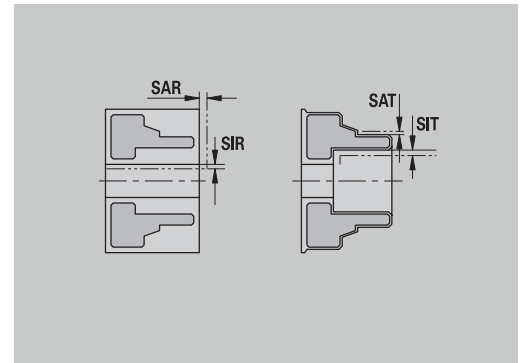
Parameter	Meaning
Front chuck edge on counter spindle	Z position of the front edge of the chuck for calculating the workpiece datum (**AWG** submode)
Jaw width on main spindle	Jaw width in Z direction for calculating the workpiece datum (**AWG** submode)
Jaw width on counter spindle	Jaw width in Z direction for calculating the workpiece datum (**AWG** submode)

Further global technology parameters

Global technology parameters

Parameter	Meaning
G14 for new units	Default setting for the axis sequence (start unit: **GWW** parameter) that is used to approach the tool change position:
	■ No axis
	■ 0: Simultaneously
	■ 1: First X, then Z
	■ 2: First Z, then X
	■ 3: Only X
	■ 4: Only Z
	■ 5: Only Y
	■ 6: Simultaneous w/ Y
Coolant for new units	Default setting for the coolant (start unit: **CLT** parameter):
	■ 0: Without coolant
	■ 1: Coolant circuit 1 on
	■ 2: Coolant circuit 2 on
Rotational direction for new units	Default for the spindle direction of rotation **MD** when creating or opening a new unit (**Tool** tab)
Conversion of the ICP contours	Type of conversion for the ICP contours
	■ 0: The calculated parameters are output
	■ 1: The programmed parameters are output

Global parameters for finished parts

Global parameters for finished parts

Parameter	Meaning
Max. inward copying angle **EKW**	Tolerance angle for recess areas, used for distinguishing turning from recessing (mtw = contour angle):
	■ **EKW** > **mtw**: Relief turn
	■ **EKW** <= **mtw**: Undefined recess (no form element)
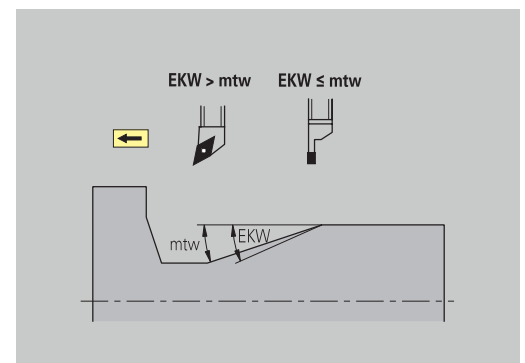
Centric predrilling

Centric predrilling – Tool selection

Tool selection

Parameter	Meaning
1st drilling diameter limit **UBD1**	■ 1st predrilling step: if **UBD1** < **DB1max** ■ Tool selection: **UBD1** <= **db1** <= **DB1max**
2nd drilling diameter limit **UBD2**	■ 2nd predrilling step: If **UBD2** < **DB2max** ■ Tool selection: **UBD2** <= **db2** <= **DB2max**

For predrilling, a maximum of three drilling steps is used:

■ 1st predrilling step (limit diameter **UBD1**)
■ 2nd predrilling step (limit diameter **UBD2**)
■ Finish-drilling step
  ■ Finish-drilling is performed with: dimin <= **UBD2**
  ■ Tool selection: **db** = **dimin**

Designations in the figures at right:

■ **db1**, **db2**: Drill diameter
■ **DB1max**: Max. inside diameter for 1st drilling step
■ **DB2max**: Max. inside diameter for 2nd drilling step
■ **dimin**: Min. inside diameter
■ **BBG** (drilling limitation elements): Contour elements intersected by **UBD1** and **UBD2**
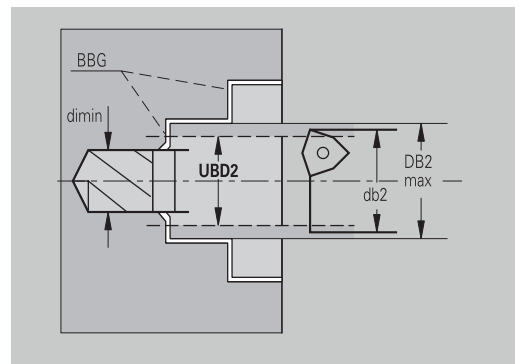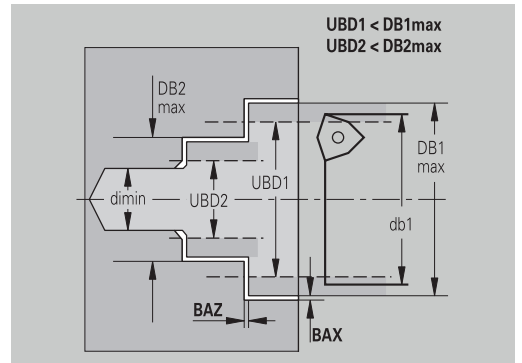
ℹ ■ **UBD1** and **UBD2** have no effect when **Centric predrilling** has been defined as main machining operation followed by **Finish-drilling** as submachining operation in the machining sequence
  Further information: smart.Turn and DIN Programming User's Manual

  ■ Prerequisite: **UBD1** > **UBD2**

  ■ **UBD2** must permit subsequent inside machining with boring bars

---

Centric predrilling – Oversizes

Oversizes

Parameter	Meaning
Point angle tolerance **SWT**	If the drilling limitation element is a diagonal element, **TURN PLUS** prefers using a twist drill with suitable point angle. If no suitable twist drill is available, an index-able-insert drill is selected for the predrilling operation. **SWT** defines the permissible point angle tolerance.
Drilling oversize – Diameter **BAX**	Machining oversize on drilling diameter (X direction – radius value)
Drilling oversize – Depth **BAZ**	Machining oversize on drilling depth (Z direction)

> **ℹ** **BAZ** is not considered if:
> - A subsequent inside finishing operation is not possible due to the small diameter
> - dimin < 2 * **UBD2** in the finish-drilling step for blind holes
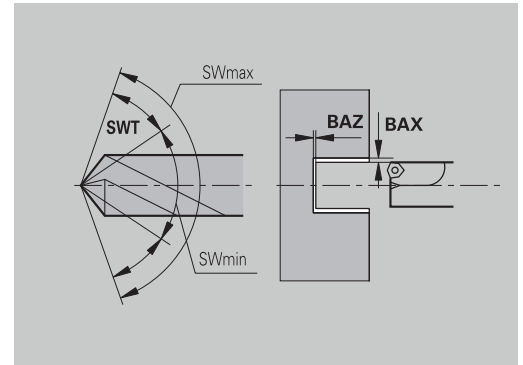
Centric predrilling – Approach and departure

Approach and departure

Parameter	Meaning
■ Traverse for predrilling **ANB** ■ Traverse for tool change **ABW**	Approach or departure strategy: ■ 1: Move simultaneously in X and Z directions ■ 2: First X, then Z direction ■ 3: First Z, then X direction
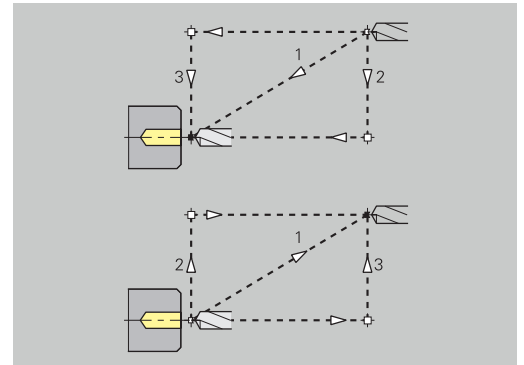
Centric predrilling – Safety clearances

Safety clearances

Parameter	Meaning
Safety clearance on workpiece blank **SAB**	
Internal safety clearance **SIB**	Retraction distance for deep-hole drilling (**B** for **G74**)
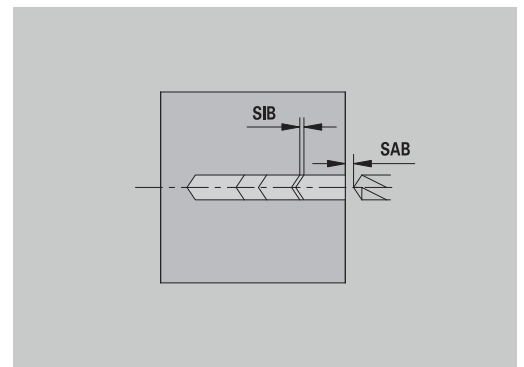
Centric predrilling – Machining

Machining

Parameter	Meaning
Drilling depth ratio **BTV**	**TURN PLUS** checks the 1st and 2nd drilling steps. The predrilling step is performed if: **BTV** <= **BT** / **dmax**
Drilling depth factor **BTF**	1st drilling depth for deep-hole drilling cycle (**G74**): **bt1** = **BTF** * **db**
Depth reduction **BTR**	Reduction for deep-hole drilling cycle (**G74**): **bt2** = **bt1** − **BTR**
Overhang length – Predrilling **ULB**	Through-drilling length
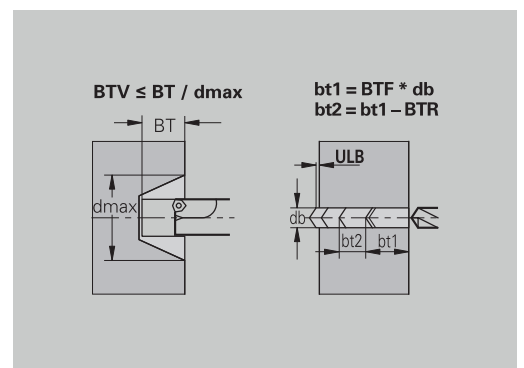
Roughing

Roughing – Tool standards

Furthermore:

- Roughing cycles are primarily executed with standard roughing tools
- Alternatively, tools that allow full-surface machining are used

Tool standards

Parameter

- Tool angle – External/Longitudinal **RALEW**
- Point angle – External/Longitudinal **RALSW**
- Tool angle – External/Transverse **RAPEW**
- Point angle – External/Transverse **RAPSW**
- Tool angle – Internal/Longitudinal **RILEW**
- Point angle – Internal/Longitudinal **RILSW**
- Tool angle – Internal/Transverse **RIPEW**
- Point angle – Internal/Transverse **RIPSW**

Roughing – Machining standards

Machining standards

Parameter

- Standard/Full-surface – External/Longitudinal **RAL**
- Standard/Full-surface – Internal/Longitudinal **RIL**
- Standard/Full-surface – External/Transverse **RAP**
- Standard/Full-surface – Internal/Transverse **RIP**

Input for **RAL**, **RIL**, **RAP**, **RIP**:

- 0: Full-surface roughing cycle, including plunge-cutting. **TURN PLUS** looks for a tool for full-surface machining
- 1: Standard roughing cycle without plunge-cutting
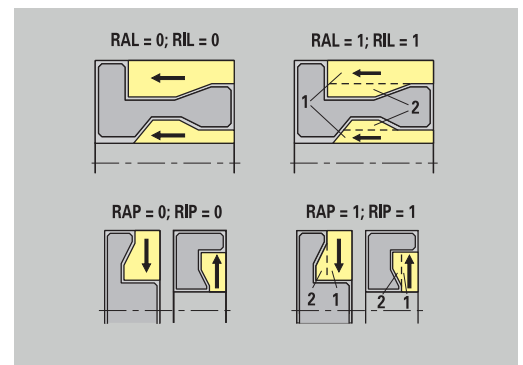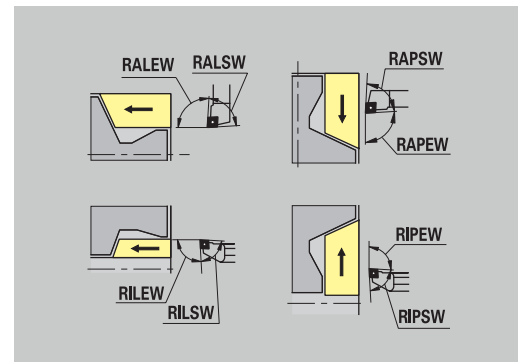
Roughing – Tool tolerances

For tool selection, the following applies:

■ Tool angle **EW**: **EW** >= **mkw** (**mkw**: ascending contour angle)

■ Tool angle **EW** and point angle **SW**: **NWmin** < (**EW**+**SW**) < **NWmax**

■ Adjacent angle **RNWT**: **RNWT** = **NWmax** − **NWmin**

Tool tolerances

Parameter	Meaning
Adjacent angle tolerance **RNWT**	Tolerance range for secondary cutting edge
Relief cutting angle **RFW**	Minimum angle differential between the contour and secondary cutting edge

Roughing – Oversizes

Oversizes

Parameter	Meaning
Type of oversize **RAA**	■ 16: Longitudinal and transverse oversizes differ – No single oversizes ■ 32: Equidistant oversize – No single oversizes ■ 144: Longitudinal and transverse oversizes differ – With single oversizes ■ 160: Equidistant oversize – With single oversizes
Equidistant or longitudinal **RLA**	Equidistant oversize or longitudinal oversize
None or transverse **RPA**	Transverse oversize
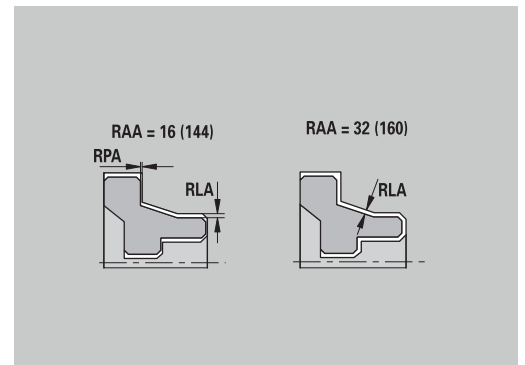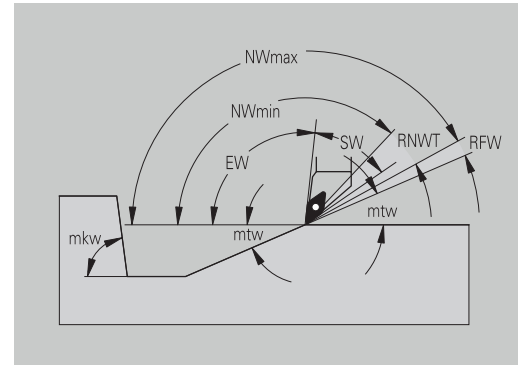
Roughing – Approach and departure

Approach and departure movements are at rapid traverse (**G0**).

Approach and departure

Parameter

- Approach – External roughing **ANRA**
- Approach – Internal roughing **ANRI**
- Departure – External roughing **ABRA**
- Departure – Internal roughing **ABRI**

Approach or departure strategy:

- 1: Move simultaneously in X and Z directions
- 2: First X, then Z direction
- 3: First Z, then X direction

Roughing – Machining analysis

TURN PLUS uses the **PLVA** and **PLVI** parameters to define whether a roughing area is to be rough-machined longitudinally or transversely.

Machining analysis

Parameter	Meaning
Transverse/longitudinal ratio – External **PLVA**	■ **PLVA** <= **AP** / **AL**: Longitudinal machining ■ **PLVA** > **AP** / **AL**: Transverse machining
Transverse/longitudinal ratio – Internal **PLVI**	■ **PLVI** <= **IP** / **IL**: Longitudinal machining ■ **PLVI** > **IP** / **IL**: Transverse machining
Minimum roughing transverse length **RMPL** (radius value)	Defines whether transverse roughing is used for the front transverse element of the outside contour of a finish part ■ **RMPL** > **l1**: Without additional transverse roughing ■ **RMPL** < **l1**: With additional transverse roughing ■ **RMPL** = **0**: Special case
Transverse angle variation **PWA**	The first front element is declared a transverse element when it is within **+PWA** and **-PWA**
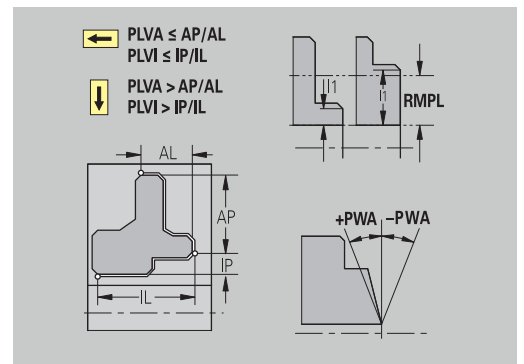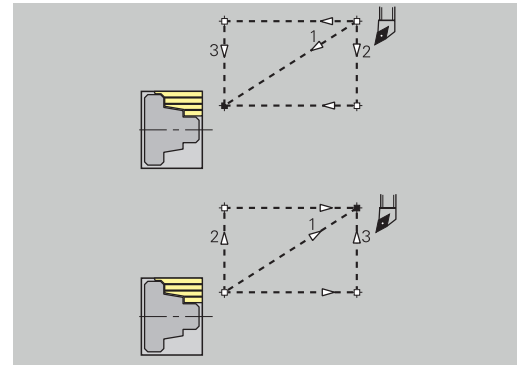
Fixed cycles

Machining analysis

Parameter	Meaning
Overhang length outside **ULA**	Relative length for external rough-machining allowing roughing beyond the target position in longitudinal direction. **ULA** is not considered when the cutting limitation is in front of or within the overhang length.
Overhang length – Inside **ULI**	■ Relative length for internal rough-machining allowing roughing beyond the target position in longitudinal direction. **ULI** is not considered when the cutting limitation is in front of or within the overhang length. ■ This parameter is used for calculating the drilling depth for centric predrilling.
Retracting length outside **RAHL**	Retracting length for smoothing variants (**H** = 1 and 2) of roughing cycles (**G810** and **G820**) for external machining (**RAHL**).
Retracting length – Inside **RIHL**	Retracting length for smoothing variants (**H** = 1 and 2) of roughing cycles (**G810** and **G820**) for internal machining (**RIHL**).
Cutting depth reduction factor **SRF**	For rough-machining with tools machining opposite to the main machining direction, the infeed value (cutting depth) is reduced. Infeed (**P**) for the roughing cycles (**G810** and **G820**): **P** = **ZT** * **SRF** (**ZT**: Infeed value from technology database)
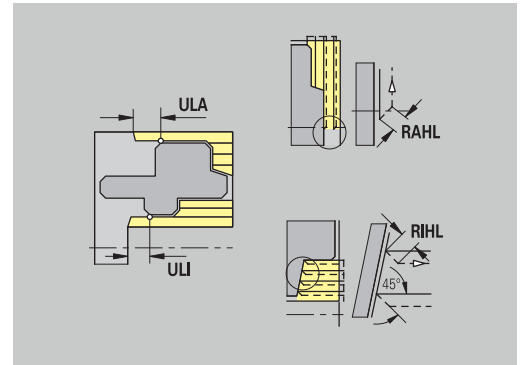
Finishing

Finishing – Tool standards

Tool selection:

- Finishing cycles are primarily executed with standard finishing tools
- If form elements such as relief turns (**form FD**) and undercuts (**form E, F, G**) cannot be machined with a standard finishing tool, one form element after the other is skipped. **TURN PLUS** starts a renewed attempt to machine the **remaining contour** with the standard tool. Subsequently, the skipped form elements are machined individually with a suitable finishing tool.

Tool standards

Parameter

- Tool angle – External/Longitudinal **FALEW**
- Point angle – Internal/Longitudinal **FILEW**
- Tool angle – External/Transverse **FAPEW**
- Point angle – Internal/Transverse **FIPEW**

Finishing – Machining standards

Machining standards

Parameter

- Standard/Full-surface – External/Longitudinal **FAL**
- Standard/Full-surface – Internal/Longitudinal **FIL**
- Standard/Full-surface – External/Transverse **FAP**
- Standard/Full-surface – Internal/Transverse **FIP**

Machining of contour areas:

- Complete: **TURN PLUS** searches for an optimum tool for machining the complete contour area
- Default:
  - The machining operation is primarily executed with standard finishing tools. Relief turns and undercuts are machined with a suitable tool
  - If the standard finishing tool cannot be used for relief turns and undercuts, **TURN PLUS** splits up the machining operation into standard machining operations and machining the form elements
  - If a division into standard machining and form-element machining is not possible, **TURN PLUS** switches to **full-surface machining**
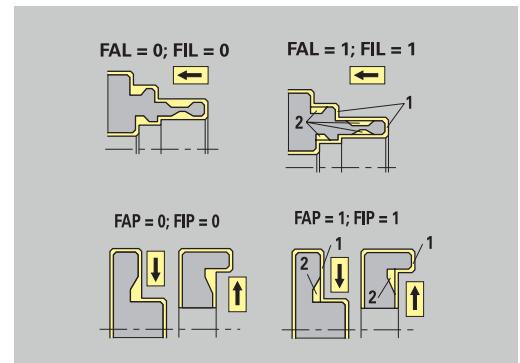
Finishing – Tool tolerances

For tool selection, the following applies:

- Tool angle **EW**: **EW** >= **mkw** (**mkw**: ascending contour angle)
- Tool angle **EW** and point angle **SW**: **NWmin** < (**EW**+**SW**) < **NWmax**
- Adjacent angle **FNWT**: **FNWT** = **NWmax** − **NWmin**

Tool tolerances

Parameter	Meaning
Adjacent angle tolerance **FNWT**	Tolerance range for secondary cutting edge
Relief cutting angle **FFW**	Minimum angle differential between the contour and secondary cutting edge
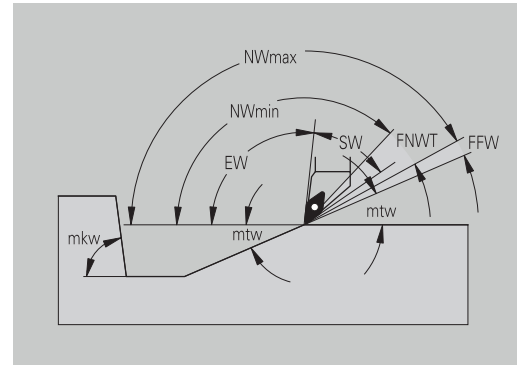
Finishing – Approach and departure

Approach and departure movements are at rapid traverse (**G0**).

Approach and departure

Parameter

- Approach – External finishing **ANFA**
- Approach – Internal finishing **ANFI**
- Departure – External finishing **ABFA**
- Departure – Internal finishing **ABFI**

Approach or departure strategy:

- 1: Move simultaneously in X and Z directions
- 2: First X, then Z direction
- 3: First Z, then X direction
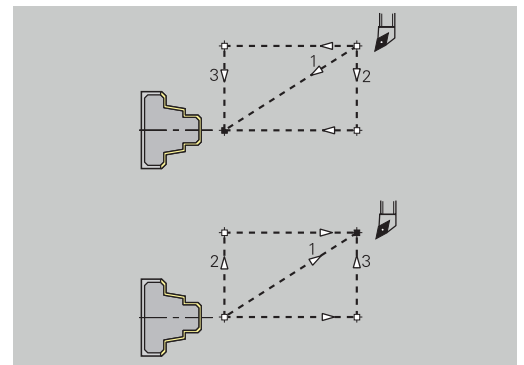
Finishing – Machining analysis

Machining analysis

Parameter	Meaning
Minimum finishing transverse length **FMPL**	**TURN PLUS** checks the front-most element of the outside contour to be finished. The following applies: ■ Without inside contour: Always with additional transverse cut ■ With inside contour – **FMPL** >= **l1**: Without additional transverse cut ■ With inside contour – **FMPL** < **l1**: With additional transverse cut
Maximum finishing cut depth **FMST**	**FMST** defines the permissible infeed depth for non-machined undercuts. The finishing cycle (**G890**) uses this parameter to determine whether undercuts (**form E, F, G**) will be machined with a contour-finishing operation. The following applies: ■ **FMST** > **ft**: With undercut machining (**ft**: undercut depth) ■ **FMST** <= **ft**: Without undercut machining
Number of revolutions for chamfer or rounding **FMUR**	The feed rate is reduced such that at least **FMUR** revolutions can be executed (evaluation: finishing cycle **G890**).
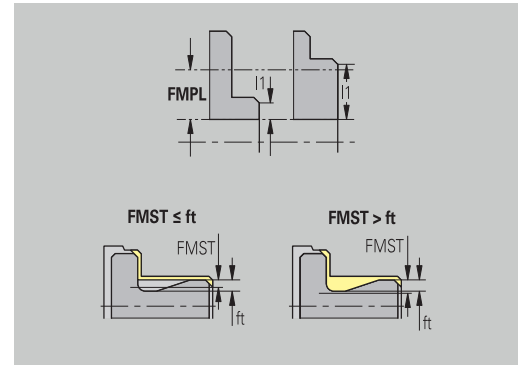
> 🛈 The following applies to FMPL:
> ■ The additional transverse cut is executed from the outside toward the inside
> ■ The **Transverse angle variation PWA** has no effect on the analysis of the transverse elements

Recessing and contour recessing

Recessing and contour recessing – Tool selection

Tool selection

Parameter	Meaning
Recessing width divisor **SBD**	If a recess base machined with a contour-recessing cycle does not contain paraxial elements but only linear elements, a suitable tool is selected using the **recessing width divisor SBD**. **SB** <= **b** / **SBD** (**SB**: width of recessing tool; **b**: width of machining area)
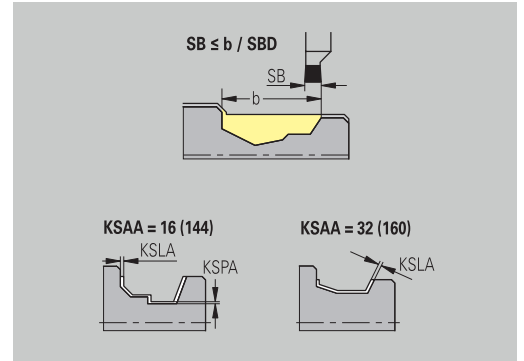
Recessing and contour recessing – Oversizes

Oversizes

Parameter	Meaning
Type of oversize **KSAA**	The recessing area to be machined can be assigned oversizes. When oversizes have been defined, a recess is first rough-machined and then finish-machined. Inputs: ■ 16: Longitudinal and transverse oversizes differ – No single oversizes ■ 32: Equidistant oversize – No single oversizes ■ 144: Longitudinal and transverse oversizes differ – With single oversizes ■ 160: Equidistant oversize – With single oversizes
Equidistant or longitudinal **KSLA**	Equidistant oversize or longitudinal oversize
None or transverse **KSPA**	Transverse oversize

> ℹ️ ■ The oversizes are accounted for when machining contour valleys with a contour-recessing operation
> ■ Standardized recesses, such as recess types **D, S, A**, are completed in one machining operation. A division into rough-machining and finish-machining is only possible in **DIN PLUS**

Recessing and contour recessing – Approach and departure

Approach and departure movements are at rapid traverse (**G0**).

Approach and departure

Parameter

- Approach – External recessing **ANESA**
- Approach – Internal recessing **ANESI**
- Departure – External recessing **ABESA**
- Departure – Internal recessing **ABESI**

- Approach – External contour recessing **ANKSA**
- Approach – Internal contour recessing **ANKSI**
- Departure – External contour recessing **ABKSA**
- Departure – Internal contour recessing **ABKSI**

Approach or departure strategy:

- 1: Move simultaneously in X and Z directions
- 2: First X, then Z direction
- 3: First Z, then X direction

Recessing and contour recessing – Machining

Evaluation: **DIN PLUS**

Machining

Parameter	Meaning
Recessing width factor **SBF**	**SBF** is used for calculating the maximum offset for recessing cycles **G860** and **G866**: **esb** = **SBF** * **SB** (**esb**: effective recessing width; **SB**: width of recessing tool)
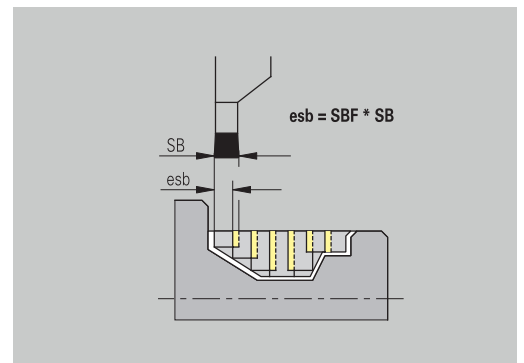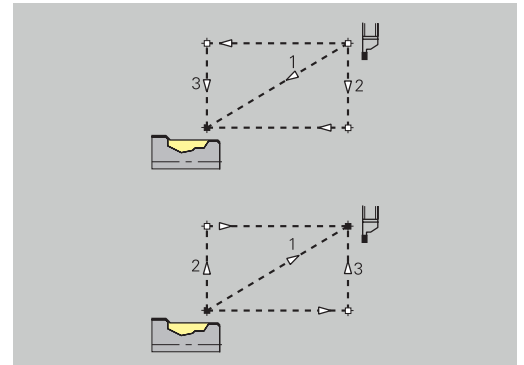
Thread cutting

Thread cutting – Approach and departure

Approach and departure movements are at rapid traverse (**G0**).

Approach and departure

Parameter

- Approach – External thread **ANGA**
- Approach – Internal thread **ANGI**
- Departure – External thread **ABGA**
- Departure – Internal thread **ABGI**

Approach or departure strategy:

- 1: Move simultaneously in X and Z directions
- 2: First X, then Z direction
- 3: First Z, then X direction

Thread cutting – Machining

Machining

Parameter	Meaning
Thread run-in length **GAL**	Run-in length before thread chamfer
Thread run-out length **GUL**	Overrun length after thread chamfer

ℹ️ **GAL** and **GUL** are automatically copied to the **Run-in lgth B** and **End. length P** thread attributes if they have not been entered as attributes.
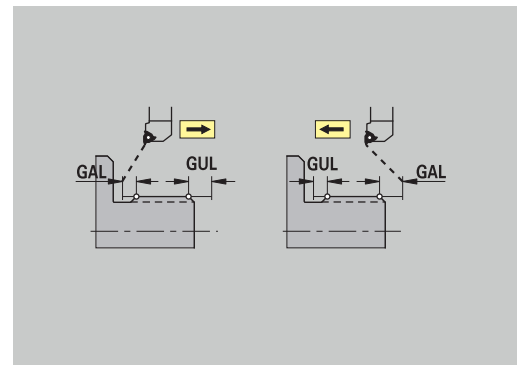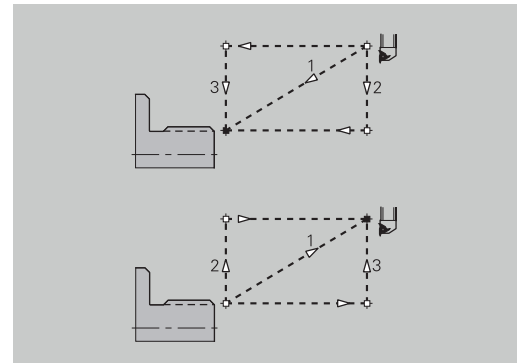
Measuring

Measuring – Measurement procedure

The measuring parameters are assigned to the fit elements as attributes.

Measurement procedure

Parameter	Meaning
Measuring loop counter **MC**	Defines the measurement/loop intervals
Measuring departing length in Z **MLZ**	Distance in Z for departure movement
Measuring departing length in X **MLX**	Distance in X for departure movement
Measuring oversize **MA**	Oversize still existing on the element to be measured

Measuring cut length **MSL**
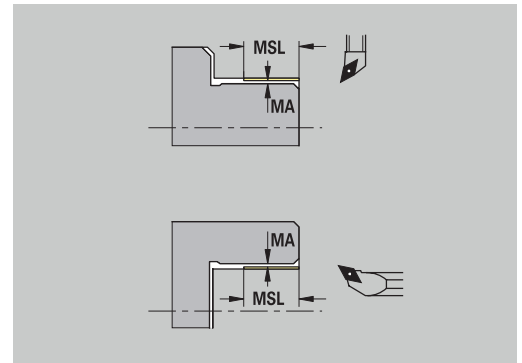
Drilling

Drilling – Approach and departure

Approach and departure movements are at rapid traverse (**G0**).

Approach and departure

Parameter

- Approach – Face **ANBS**
- Approach – Lateral surface **ANBM**
- Departure – Face **ABGA**
- Departure – Lateral surface **ABBM**

Approach or departure strategy:

- 1: Move simultaneously in X and Z directions
- 2: First X, then Z direction
- 3: First Z, then X direction

Drilling – Safety clearances

Safety clearances

Parameter	Meaning
Internal safety clearance **SIBC**	Retraction distance for deep-hole drilling (**B** for **G74**)
Driven drills **SBC**	Safety clearance for driven tools on face and lateral surface
Stationary drills **SBCF**	Safety clearance on face and lateral surface for tools that are not driven
Driven taps **SGC**	Safety clearance for driven tools on face and lateral surface
Stationary taps **SGCF**	Safety clearance on face and lateral surface for tools that are not driven
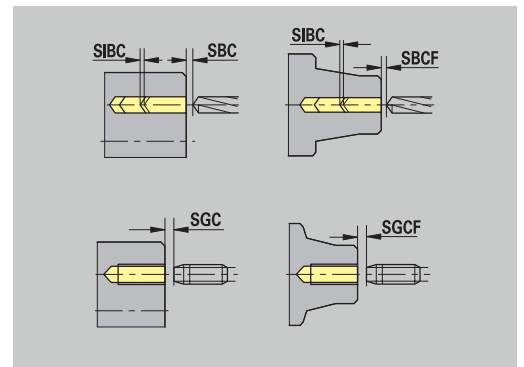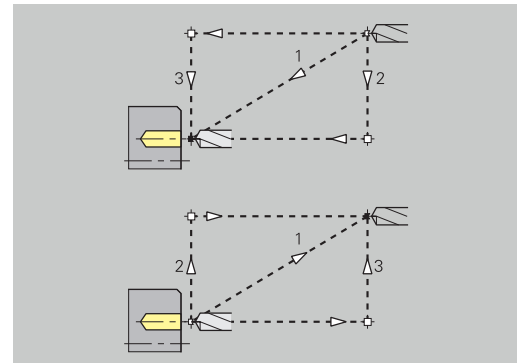
Drilling – Machining

The parameters apply to drilling with deep-hole drilling cycle (**G74**).

Machining

Parameter	Meaning
Drilling depth factor **BTFC**	1st drilling depth: **bt1** = **BTFC** * **db** (**db**: Drill diameter)
Depth reduction **BTRC**	2nd drilling depth: **bt2** = **bt1** − **BTRC** The subsequent drilling steps are reduced accordingly
Diameter tolerance for drill **BDT**	For selecting the desired drill (centering drills, countersinks, stepped drill, taper reamers). ■ Drilling diameter: **DBmax** = **BDT** + **d** (**DBmax**: Maximum drilling diameter) ■ Tool selection: **DBmax** > **DB** > **d**
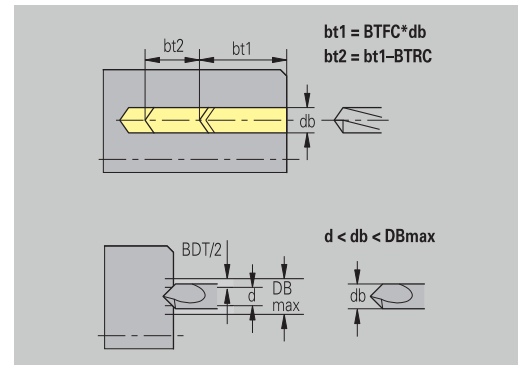
Milling

Milling – Oversizes

Oversizes

Parameter

Oversize in milling direction **MEA**

Oversize in infeed direction **MZA**

Milling – Approach and departure

Approach and departure movements are at rapid traverse (**G0**).

Approach and departure

Parameter

- Approach – Face **ANMS**
- Approach – Lateral surface **ANMM**
- Departure – Face **ABMS**
- Departure – Lateral surface **ABMM**

Approach or departure strategy:

- 1: Move simultaneously in X and Z directions
- 2: First X, then Z direction
- 3: First Z, then X direction

Milling – Safety clearances

Safety clearances

| Parameter	Meaning
Safety clearance in infeed direction **SMZ**	Distance between starting position and top edge of object to be milled
Safety clearance in milling direction **SME**	Distance between milling contour and edge of milling cutter
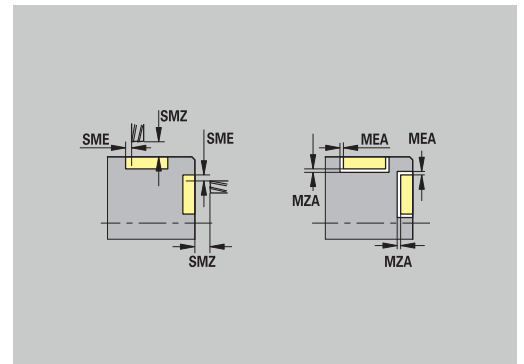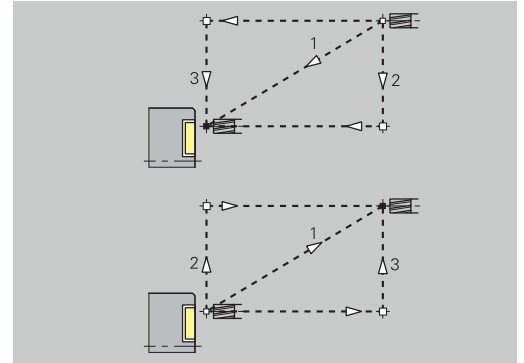
Milling – feed rate

The parameters apply to trochoidal milling and area milling.

Feed rate

Parameter	Meaning
Infeed rate factor (trochoidal milling)	Factor by which the feed rate deviates from the nominal value during plunging
Feed rate factor (trochoidal milling)	Factor by which the feed rate deviates from the nominal value during machining
Infeed rate factor (area milling)	Factor by which the feed rate deviates from the nominal value during plunging

Milling – machining

The parameters apply to trochoidal milling and area milling.

Use the **RC** parameter to define trochoidal milling in ICP programming.

Further information: "TURN PLUS attributes", Page 494

Machining

Parameter	Meaning
Plunge angle	Angle of the plunge-cut during reciprocating or helical plunge
Helix diameter factor	Factor by which the plunging helix is greater than the cutter diameter
Overlap factor (trochoidal milling)	Proportional factor of the overlap factor
Retraction feed rate (trochoidal milling)	Feed rate when the tool is not engaged during the trochoidal milling
Retraction distance (trochoidal milling)	Value in millimeters by which the tool lifts off of the surface during retraction.
Overlap factor (area milling)	Proportional factor of the overlap factor
Overrun factor (area milling)	Factor by which the tool mills beyond the surface

11.3 Transfer submode

Transfer submode is used for data backup and data exchange via networks or USB devices. Speaking of **files** in the following, means programs, parameters, or tool data.

The following file types can be transferred:

- Programs (cycle programs, smart.Turn programs, DIN main programs and DIN subprograms, ICP contour descriptions)
- Parameters
- Tool data

> **i** Screenshots (taken with the **PRT SC** key) can be transferred from the **Service** area.

Data backup

HEIDENHAIN recommends backing up the tool data and programs created on the control on an external device at regular intervals.

You should also back up the parameters. Since the parameters are not changed very often, however, you only need to back up the parameters from time to time, as required.

Data exchange with TNCremo

HEIDENHAIN offers the PC program TNCremo to complement the control. This program enables you to access the control data from a PC.

External access

> **i** The machine tool builder can configure the external access options. Refer to your machine manual.

The **External access** soft key allows you to permit or restrict access via the LSV2 interface.

To permit external access:

▶ Select the **Organization** operating mode

▶ Set the **External access** soft key to **ON**

> The control will then permit data access through the LSV2 interface

To restrict external access:

▶ Select the **Organization** operating mode

▶ Set the **External access** soft key to **OFF**

> The control will then disable access through the LSV2 interface

Connections

You can establish connections over the network (Ethernet) or with a USB storage device. Data is transferred over the **Ethernet** or **USB** interface.

- Network (via Ethernet): The control supports **SMB** networks (Server Message Block, WINDOWS) and **NFS** networks (Network File Service)
  Further information: "Ethernet interface ", Page 719
- USB data media can be connected directly to the control. The control uses only the first partition of a USB storage device
  Further information: "USB connection", Page 663

NOTICE

Danger of collision!
Caution: Data may be lost!

Unprotected or insufficiently configured networks facilitate unauthorized and insecure access to control data. As a consequence, NC programs and machine settings may be changed or deleted. This results in a risk of data loss and in an increased danger of collision!

- ▶ Enable only authorized persons to access the control over the network
- ▶ If required, deactivate external access in the **Organization** operating mode
- ▶ Alternative: Write-protect data, where necessary

HEIDENHAIN also recommends backing up the data at regular intervals.

ⓘ You can also create new folders on a connected USB storage device or network drive. Press the **Create transfer folder** soft key and enter a name for the folder.

The control displays all active connections in a selection window. If a folder contains subfolders, you can open and select these subfolders.

The network configuration is provided in the HEROS menu.

Further information: "Network settings window", Page 720

USB connection

To establish a USB connection:

▶ Select **Organization** operating mode

▶ Connect the USB storage device to the USB interface of the control

TRANSFER

▶ Press the **Transfer** soft key (login required)

Con-
nections

▶ Press the **Connections** soft key

> The USB storage device is displayed.

▶ Use the soft keys to disconnect or reconnect USB storage devices

ℹ In principle, it should be possible to connect most USB devices to the control. It may happen that a USB device is not identified correctly by the control, for example when the cable between the control panel and the main computer is very long. In such cases, use another USB device.

Soft keys for USB connection

Soft key	Meaning
Create folder	Create a folder of the specified name on the USB storage device
	Disconnect the USB storage device and prepare the device for removal
Free external	Allow access to files that have not been properly saved in a project folder
Selection	Select the project folder that you selected before with the arrow keys
Back	Return to the Transfer soft-key menu

Data transfer options

The control manages DIN programs, DIN subprograms, cycle programs, and **ICP contours** in different directories. When you select the **Program group**, the control automatically switches to the applicable directory.

Parameters and tool data are stored under the file name entered for backup name and saved to a ZIP file located in the **para** or **tool** folder on the control. You can then send this backup file to a project folder in the remote station.

> **ⓘ**   ■ If program files are opened in another operating mode, they are not overwritten
>   ■ You can transfer tool data and parameters only if no program has been started in **Program run** submode

The following transfer functions are available:

- **Programs**: Transmit and receive files
- **Parameter backup**: Create, transmit, and receive parameter backups
- **Parameter restore**: Restore the parameter backup files
- **Tools backup**: Create, transmit, and receive tool backups
- **Restore tools**: Restore the tool backup files
- **Service data**: Create and transmit service data
- Make **data backup**: Save all data in a project folder
- **Free external**: Freely select program files on a USB storage device
- **Misc. functions**: Import cycle programs and DIN programs from the MANUALplus 4110 and import tool data from the CNC PILOT 4290

Folder structure – Storage of files

Folder	File types
\dxf	Drawings in DXF format
\gtb	Machining sequences (**TURN PLUS**)
\gti	ICP contour definitions: ■ ***.gmi** (turning contour) ■ ***.gmr** (workpiece-blank contour) ■ ***.gms** (C axis, face) ■ ***.gmm** (C axis, lateral surface)
\gtz	Cycle programs (**Teach-in** submode) ***.gmz**
\ncps	DIN programs (**smart.Turn** mode of operation): ■ ***.nc** (main programs) ■ ***.ncs** (subprograms)
\para	Parameter backup files PA_*.zip (parameters)
\table	Parameter backup files TA*.zip (tables)
\tool	Tool backup files TO*.zip (tool and technology data)
\pictures	Image files for subprograms ***.bmp**, ***.png** or ***.jpg**
\data	Service files Service*.zip

Transfer folder

The control exchanges data with an external storage device by accessing transfer folders created in the storage device. Files in transfer folders are organized using the same folder structure as on the control.

Transfer folders can only be used directly in the selected network path or in the root directory of the USB storage device.

Transferring programs (files)

Selecting a program group

ⓘ Screenshots (taken with the **PRT SC** key) can be transferred from the **Service** area.

To select a program group:

▶ Select **Organization** operating mode

▶ Press the **Transfer** soft key (login required)

TRANSFER

▶ Press the **Connections** soft key

Con-nections

▶ Select a project folder and then press the **Selection** soft key (USB)

Selection

▶ Press the **Back** soft key to return to data selection

Back

▶ Switch to program transfer

Programs

▶ Open a selection of program types

Program selection

▶ Press the DIN programs (or other program types) soft key to activate transfer

DIN programs

ⓘ With the transfer of automatic jobs, the control automatically transfers the selected jobs with all main programs and the subprograms contained in these.
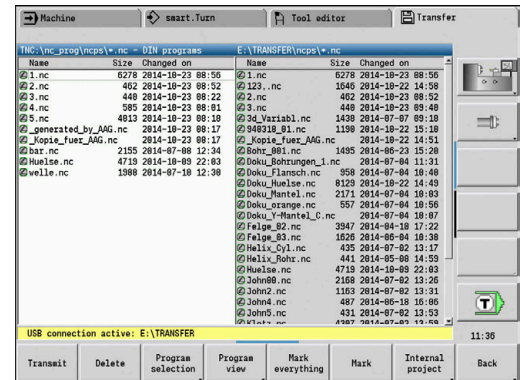
Soft keys for program group selection

Soft key	Meaning
DIN programs	***.nc**: DIN and smart.Turn main programs. **Transfer** submode scans programs for subprograms and asks whether you want to transfer them as well.
DIN subprogram	***.ncs**: DIN and smart.Turn subprograms. Help graphics are also transferred.
Cycle programs	***.gmz**: Cycle programs. **Transfer** submode scans the programs for subprograms and **ICP contours**, and asks whether you want to transfer them as well.
ICP contours	**ICP contours** for cycle programs: ■ ***.gmi** (turning contour) ■ ***.gmr** (workpiece-blank contour) ■ ***.gms** (C axis, face) ■ ***.gmm** (C axis, lateral surface)
Next file type	Switch between possible file types. You can also select automatic jobs here.
Free external	Select program files on a USB storage device without using a project folder.
File mask	Mask the file names within the selected program group.

Selecting the program

In the window on the left, the control shows a file list of the control. The files of the remote station are displayed in the window on the right when a connection is established. With the arrow keys you can switch back and forth between the two windows.

When selecting a program, place the cursor on the desired program and press the **Mark** soft key. You can also select all programs with the **Mark everything** soft key.

Marked programs are highlighted in color. To unmark a program, simply mark the program again.

The control displays the file size and the time of the last modification of the program in the list if permitted by the length of the file name.

With DIN programs/subprograms, you can also view the NC program with the **Program view** soft key.

Press the **Transmit** or **Receive** soft key to start transferring the files.

During transfer, the control displays the following information in a transfer window:

- The name of the program which is currently being transferred
- If a file already exists in the target path, the control asks whether you want to overwrite the file. You can then activate overwriting for only that file or for all files in the transfer

If the control detects during transfer that the transferred files are linked to other files (subprograms, **ICP contours**), a dialog opens where you can select the linked files for transfer.
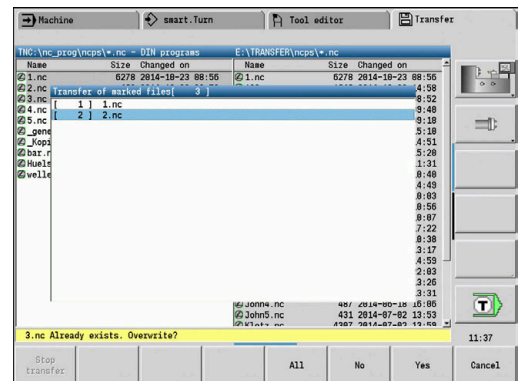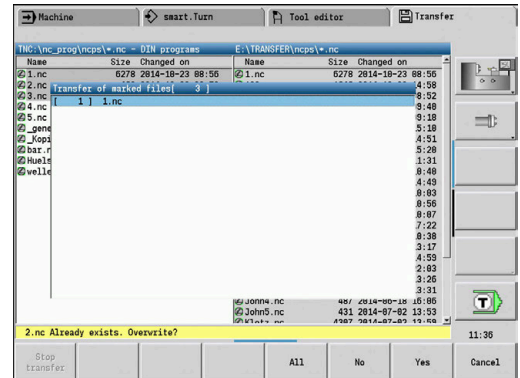
Soft keys for program selection

Soft key	Meaning
Mark everything	Mark all files in the current window
Mark	Mark or unmark the file at the cursor position and move the cursor down one position
Program view	Open a DIN main program or subprogram for reading

Transferring project files

If you wish to transfer files from a project, the **Project** soft key enables you to open the project management of the control and select the corresponding project.

Further information: "Project management", Page 191

> ℹ The **Internal project** soft key enables you to manage projects and transfer entire project folders.
> **Further information:** "Project management", Page 191

Transferring parameters

Within the Parameters menu, the following functions are available to you:

- **Back up parameters**: The parameters are saved as ZIP files and stored in the control, see "Backing up parameters", Page 670
- **Transmit or receive parameter backup files**: The ZIP files are transferred to an external drive or to the control, see "Transmitting or receiving the parameter backup files", Page 670
- **Restore parameters**: Restore the saved backup files on the control (only for machine manufacturers).
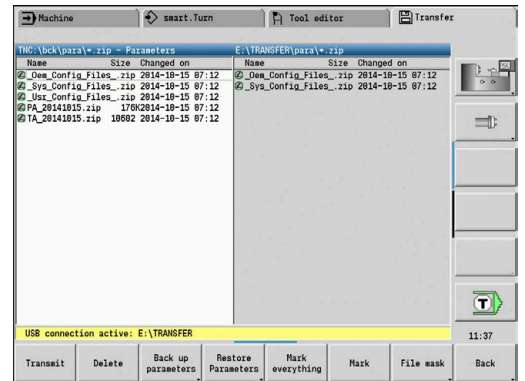
Parameters menu

To open the Parameters menu:

▶ Select **Organization** operating mode

▶ Press the **Transfer** soft key

▶ Press the **Parameters** soft key

Parameters soft key

Soft key	Meaning
Transmit	Transmit all marked files from the control to an external drive
Receive	Receive all files marked on the external drive
Delete	Delete all marked files (code number 123)
Back up parameters	Save parameter backup files as ZIP folder (code number 123)
Restore Parameters	Restore data from the selected backup file on the control (only for machine manufacturers)
Mark everything	Mark all files in the current window
Mark	Mark the file at the cursor position or cancel the marking
File mask	Open the file mask

Backing up parameters

> ⓘ   The "Back up parameters" function requires code number 123.
>
> You can also create a parameter backup without connecting to an external drive.

With a parameter backup, you save all the parameters and tables of the control except for tool and technology data.

To create a parameter backup, proceed as follows:

Back up parameters	▶ Press the **Back up parameters** soft key
	> The control opens a pop-up window.
	▶ Define a name for the backup in the **Backup name** field
Start backup	▶ Press the **Start backup** soft key
	> The control creates a backup copy of the parameters and tables in separate ZIP files and saves the files under the following path:

- Parameters: TNC:/bck/para/PA_*.zip
- Tables: TNC:/bck/table/TA_*.zip

Transmitting or receiving the parameter backup files

If the control is connected with an external data medium, you can transmit and receive parameter backup files.

To transmit parameter backup files, proceed as follows:

▶ Mark the file in the file window on the left

Transmit	▶ Press the **Transmit** soft key
	> The control transfers the marked files to the external drive.

To receive parameter backup files, proceed as follows:

▶ Mark the file in the file window on the right

Receive	▶ Press the **Receive** soft key
	> The control transfers the marked files from the external drive to the control.

Transferring tool data

Within the Tools menu, the following functions are available to you:

- **Back up tools**: The tool data are saved as a ZIP file and stored in the control, see "Creating a tool backup", Page 672
- **Transmit or receive tool backup files**: Transfer the ZIP files to an external drive or to the control, see "Transmitting or receiving the tool backup files", Page 672
- **Restore tools**: Restore the saved backup files on the control, see "Restoring tools", Page 673

> ⓘ   The "Back up tools" and "Restore tools" functions require code number 123.
>
> You can also create and restore tool backups without connecting to an external drive.
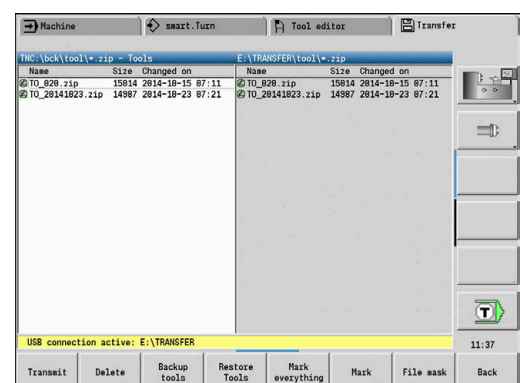
Tools menu

To open the Tools menu:

⊞

▶ Select **Organization** operating mode

TRANSFER

▶ Press the **Transfer** soft key
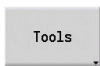
Tools

▶ Press the **Tools** soft key

Soft keys for tools

Soft key	Meaning
Transmit	Transmit all marked files from the control to an external drive
Receive	Receive all files marked on the external drive
Delete	Delete all marked files (code number 123)
Backup tools	Save tool backup files as ZIP file (code number 123)
Restore Tools	Restore data from the selected backup file on the control (code number 123)
Mark everything	Mark all files in the current window
Mark	Mark the file at the cursor position or cancel the marking After marking, the cursor is at the next position.
File mask	Select the ZIP or HTT file type The tool data can also be transferred directly as a HTT file (e.g. from a tool presetter).

Creating a tool backup

> ℹ️ The "Back up tools" and "Restore tools" functions require code number 123.

When creating a tool backup, you can save the data of all tools or of selected tools. Select the tools from the tools list or from the tools currently in the machine.

To create a tools backup, proceed as follows:

Backup tools

- ▶ Press the **Backup tools** soft key
- > The control opens a selection window.
- ▶ Define a name for the backup in the **Backup name** field
- ▶ Define the content of the backup
  - ▪ **Tools**
  - ▪ **Tool texts**
  - ▪ **Technology data**
  - ▪ **Touch probe**
  - ▪ **Tool holder**

Tool List

- ▶ Press the **Tool list** soft key
- > The control displays all defined tools in the tool list.

Tools in machine

- ▶ Alternative: Press the **Tools in machine** soft key
- > The control displays the tools currently in the machine.

Mark

- ▶ Press the **Mark** soft key to select the desired tools

Confirm selection

- ▶ Press the **Confirm selection** soft key
- > The control displays the selected tools in the **Tool selection** field.

Start backup

- ▶ Press the **Start backup** soft key
- > The control creates the backup as a ZIP file and saves the backup under the path: TNC:/bck/tool/TO_*.zip

Transmitting or receiving the tool backup files

If the control is connected with an external data medium, you can transmit and receive tool backup files.

To transmit the backup files of the tools, proceed as follows:

- ▶ Mark the file in the file window on the left

Transmit

- ▶ Press the **Transmit** soft key
- > The control transfers the marked files to the external drive.

To receive tool backup files, proceed as follows:

- ▶ Mark the file in the file window on the right

Receive

- ▶ Press the **Receive** soft key
- > The control transfers the marked files from the external drive to the control.

Restoring tools

i	The "Back up tools" and "Restore tools" functions require code number 123.

You can restore tool backup files from an external data medium or from the control. You can select the tools and data to be taken over.

To restore a backup file, proceed as follows:

▶ Select the backup file to be restored

[Restore Tools] 　 ▶ Press the **Restore Tools** soft key

　 > The control opens a selection window.

　 ▶ Define the contents to be restored

　 ■ **Tools**

　 ■ **Tool texts**

　 ■ **Technology data**

　 ■ **Touch probe**

　 ■ **Tool holder**

[Tool List] 　 ▶ Press the **Tool list** soft key, if required

　 > The control displays all tools of the backup.

[Mark] 　 ▶ Press the **Mark** soft key to select the desired tools as needed

[Confirm selection] 　 ▶ Press the **Confirm selection** soft key, if applicable

　 > The control also displays the selected tools in the **Tool selection** field.

[Start restoring] 　 ▶ Press the **Start restoring** soft key

　 > The control now loads the selected data.

Creating service files

Service files contain various log files used by the service department for troubleshooting. All important information is summarized in a services file data record as a zip file.

Path and file names of the backup files: \data\SERVICEx.zip ("x" stands for a consecutive number)

The control always generates the service file with the number **1**. Already existing files are renamed to the numbers **2 to 5**. An existing file with the number **5** is deleted.

Service files are saved in the following steps:

■ **Creating service files**: The information is summarized in a ZIP file and stored in the control

■ **Transmitting service files**

Selecting service files

You can also create service files without connecting to an external storage device.

To create service files:

[💾] 　 ▶ Select **Organization** operating mode

TRANSFER	▶ Press the **Transfer** soft key (login required)
Service	▶ Press the **Service** soft key

Soft keys for transferring service files

Soft key	Meaning
Transmit	Transmit all marked files from the control to the remote station
Delete	Delete all marked files after a confirmation prompt (login required)
Select TNC:	Select the TNC main directory, e.g. when if you want to transfer previously created screenshots (by pressing the **PRT SC** key)
Mark everything	Mark all files in the current window
Mark	Mark or unmark the file at the cursor position and move the cursor down one position
Create Service files	Create a service files data record as a .zip file

Saving service files

To save service files:

Create Service files	▶ Press the **Create files** soft key
	▶ Enter a file name under which you want to save the service file
Save	▶ Press the **Save** soft key

Creating a data backup file

Creating a data backup file involves the following steps:

- Copying the program files to the transfer folder
  - NC main programs
  - NC subprograms (with graphics)
  - Cycle programs
  - ICP contours
- Creating a parameter backup and copying all backup files from the **para** and **table** folders to the project folder. (PA_Backup.zip, TA_Backup.zip)
- Creating a tool backup file and copying all tool backup files from the **tool** folder to the project folder (TO_Backup.zip)
- Service files are **not** created and copied

Selecting a data backup file

You can also create a data backup file without connecting to an external storage device.

To create a data backup file:

▶ Select **Organization** operating mode

▶ Press the **Transfer** soft key (login required)

▶ Press the **Data Backup** soft key

Soft keys for transferring data backup files

Soft key	Meaning
Start backup	Start backing up the data to a complete transfer folder

> ■ Existing files will be overwritten without a confirmation prompt
> ■ To cancel the data backup, press the **Cancel** soft key. The backup process stops as soon as the currently running partial backup is completed

Importing NC programs from predecessor controls

The program formats of the predecessor controls MANUALplus 4110 and CNC PILOT 4290 differ from the format of the MANUALplus 620 and the CNC PILOT 640. However, you can use the program converter to adapt programs of the predecessor control to the new control. This converter is a component of your control. The converter completes the required adaptations as automatically as possible.

Overview of convertible NC programs:

- MANUALplus 4110
  - Cycle programs
  - ICP contour descriptions
  - DIN programs
- CNC PILOT 4290: DIN-PLUS programs

TURN PLUS programs of the CNC PILOT 4290 can **not** be converted.

Importing NC programs from the connected data medium

To import NC programs:

[icon]	▶ Select **Organization** operating mode
TRANSFER	▶ Press the **Transfer** soft key (login required)
Misc. functions	▶ Open the menu with the **Misc. functions** soft key
Import functions	▶ Open the menu with the **Import functions** soft key
Cycle programs	▶ Selection of cycle programs or ICP contours of the MANUALplus 4110 (*.gtz)
DIN programs	▶ Alternative: Selection of DIN programs of the MANUALplus 4110 (*.nc/ *.ncs)
DIN programs	▶ Alternative: Selection of programs of the CNC PILOT 4290 (*.nc/ *.ncs)
ENT	▶ Use the cursor keys to select the folder, then press the **ENT** key to switch to the right window
	▶ With the cursor keys, select the NC program to be converted
Mark all	▶ Alternative: Mark all NC programs
Start import	▶ Start the import filter for converting the program(s) to the format of your control
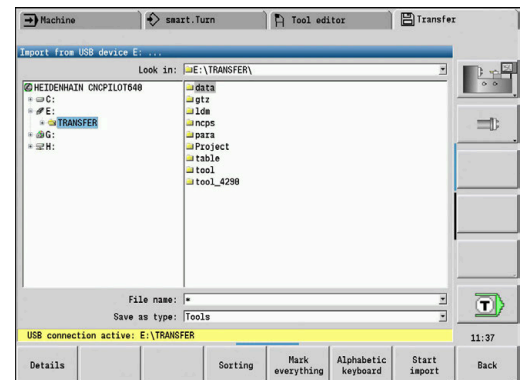
> **i** The names of imported cycle programs, ICP contour descriptions, DIN programs and DIN subprograms are given the prefix **CONV_…**.
>
> In addition, the control adapts the extension and imports the NC programs to the correct folders.

Converting cycle programs

MANUALplus 4110 and MANUALplus 620 or CNC PILOT 640 have different solutions for tool management, technology data, etc.

Also, the cycles of the MANUALplus 620 or CNC PILOT 640 have more parameters than those of the MANUALplus 4110.

Please note the following:

- **Tool call**: The loading of the tool number depends on whether the program is a **multifix program** (2-digit tool number) or **turret program** (4-digit tool number)
  - 2-digit tool number: The tool number is loaded as **ID** and entered as the tool number **T1**
  - 4-digit tool number (Tddpp): The first two digits of the tool number (dd) are loaded as **ID** and the last two (pp) as **T**
- **Approaching the tool change position**: In **Tool change point G14**, the converter enters the **no axis** setting. This parameter is not used in the MANUALplus 4110
- **Safety clearance**: The safety clearances defined in the **General Settings** parameter are entered by the converter into the **Safety clearance G47**, ... **SCI**, ... **SCK** fields
- **M functions** are left unchanged
- **Calling ICP contours**: When ICP contours are called, the converter prefixes **CONV_...** to the name
- **Calling DIN cycles**: When a DIN cycle is called, the converter prefixes **CONV_...** to the name

---

NOTICE

Danger of collision!

Converted NC programs may have incorrectly converted contents (machine-dependent) or nonconvertible contents. Danger of collision during machining!

▶ Adapt converted NC programs to the current control
▶ Test the NC program in **Simulation** submode, using the graphic displayed there

---

Converting DIN programs

When converting DIN/ISO programs, the different solutions for tool management, technology data, etc. must be taken into account, as well as contour description and variable programming.

Remember the following when converting DIN/ISO programs of the MANUALplus 4110:

- **Tool call**: The loading of the tool number depends on whether the program is a **multifix program** (2-digit tool number) or **turret program** (4-digit tool number)
- **Moving to the tool change point**: In **tool change point G14**, the converter enters the **no axis** setting. This parameter is not used in the MANUALplus 4110
- **Workpiece-blank definition**: A **G20** and **G21** workpiece-blank definition of the MANUALplus 4110 becomes an **AUX. BLANK** on your control
- **Contour descriptions**: In MANUALplus 4110 programs, the machining cycles are followed by the contour description. During conversion, the contour description is converted to an **AUX. BLANK**. The associated cycle in the **MACHINING** section then refers to this auxiliary contour
- **Variable programming**: Variable accesses to tool data, machine dimensions, **D** compensation values, parameter data, and events cannot be converted. These program sequences have to be adapted
- **M functions** are left unchanged
- **Inches or metric**: The converter cannot detect the unit of measure of the MANUALplus 4110 program Consequently, no unit of measure is entered in the target program. This has to be completed by the user

Remember the following when converting DIN programs of the CNC PILOT 4290:

- **Tool call** (**T** commands of the **TURRET** section):
  - **T** commands containing a reference to the tool database are left unchanged (example: T1 ID"342-300.1")
  - **T** commands containing tool data cannot be converted
- **Variable programming**: Variable accesses to tool data, machine dimensions, **D** compensation values, parameter data, and events cannot be converted. These program sequences have to be adapted
- **M functions** are left unchanged
- **Names of external subprograms**: When an external subprogram is called, the converter prefixes **CONV_...** to the name

> **i** If the DIN program contains nonconvertible elements, the corresponding NC block is saved as a comment. The word **WARNING** is inserted in front of this comment. Depending on the situation, the nonconvertible command is taken into the comment line, or the nonconvertible NC block follows the comment.

NOTICE

Danger of collision!

Converted NC programs may have incorrectly converted contents (machine-dependent) or nonconvertible contents. Danger of collision during machining!

▶ Adapt converted NC programs to the current control
▶ Test the NC program in **Simulation** submode, using the graphic displayed there

---

Importing tool data from the CNC PILOT 4290

The format of the tool list of the CNC PILOT 4290 differs from the format of the MANUALplus 620 and CNC PILOT 640. You can use the program converter to adapt tool data to the new control.
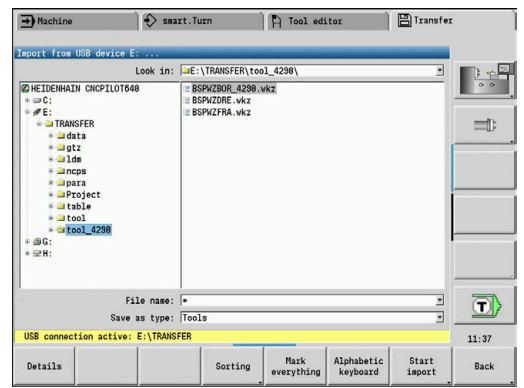
Importing tool data from the connected data medium

To import tool data:

🖫	▶ Select **Organization** operating mode
TRANSFER	▶ Press the **Transfer** soft key (login required)
Misc. functions	▶ Open the menu with the **Misc. functions** soft key
Import functions	▶ Open the menu with the **Import functions** soft key
Tools	▶ Press the **Tools** soft key
ENT	▶ Use the cursor keys to select the folder, then press the **ENT** key to switch to the right window
↓	▶ Use the arrow keys to select the tool data
Mark all	▶ Alternative: Mark all tool data
Start import	▶ Start the import filter for the conversion

The control generates a table named CONV_*.HTT for each imported file. You can load this table with the Restore function by setting the file mask to the ***.htt** file type.

11.4 Service pack

If changes or enhancements are required in the control software, your machine tool builder will provide you with a service pack. The service pack is usually installed with the aid of a USB memory stick (1 GB or larger). The software required for the service pack is compressed in the **setup.zip** file. This file is saved on the USB stick.

Installing a service pack

The control is shut down during the installation of the service pack. So end any NC program editing, etc. before you begin installation.

> ### NOTICE
>
> **Caution: Data may be lost!**
>
> The control does not back up the data automatically before installing a service pack. Power failures or other problems may interfere with the installation of a service pack. As a consequence, data may be irreversibly damaged or deleted.
>
> ▶ Save the data through a backup before installing the service pack

To install a service pack:

▶ Connect a USB stick

▶ Switch to **Organization** operating mode

▶ Press the **Key** soft key

▶ Enter the code number 231019

▶ Press the **UPDATE DATA** soft key (if the soft key is not shown, switch the soft-key menu)

▶ Press the **LOAD** soft key

▶ Press the **PATH** soft key to select the directory in the left window

▶ Press the **FILES** soft key to select the file in the right window

▶ Place the cursor on the **setup.zip** file

▶ Press the **SELECT** soft key

> The control checks whether the service pack can be used for the current software version of the control

▶ Confirm the **Do you really want to switch off?** prompt

> The update program starts

▶ Select the desired language (English or German)

▶ Run the update

> The control restarts automatically after the update has been completed.
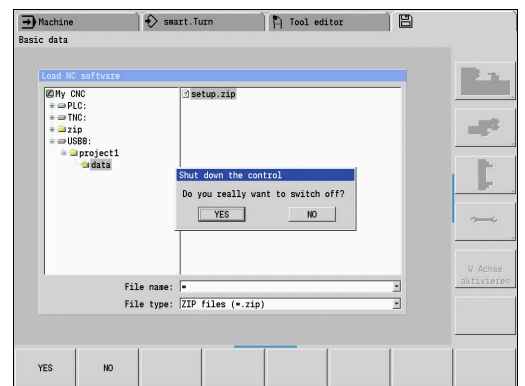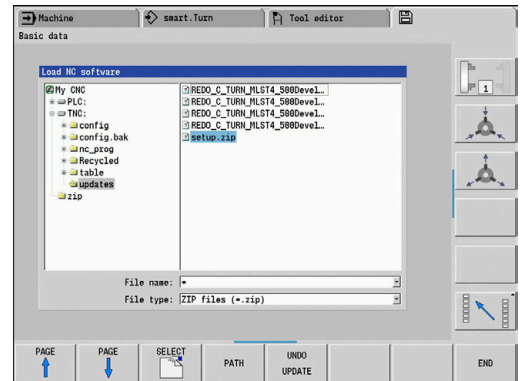
12

HEROS Functions

12.1 Remote Desktop Manager (option 133)

Introduction

Remote Desktop Manager enables you to display external computer units on the control screen that are connected via Ethernet, and to operate them via the control. You can also start programs specifically under HEROS or display web pages of an external server.

HEIDENHAIN offers you the IPC 6641 as a Windows computer. With the IPC 6641 Windows computer, you can start and operate Windows-based applications directly from within the control.

- **Windows Terminal Service (RemoteFX)**: Displays the desktop of a remote Windows computer on the control
- **VNC**: Connection to an external computer. Displays the desktop of a remote Windows, Apple, or Unix computer on the control
- **Switch-off/restart of a computer**: Configure automatic shutdown of a Windows computer
- **World Wide Web**: Available only to authorized specialists
- **SSH**: Available only to authorized specialists
- **XDMCP**: Available only to authorized specialists
- **User-defined connection**: Available only to authorized specialists

> **ⓘ** HEIDENHAIN assures a functioning connection between HEROS 5 and the IPC 6641.
>
> No guarantee is given for other combinations and connections.

Configuring connections – Windows Terminal Service (RemoteFX)

Configuring an external computer

ℹ️ You do not need additional software for your external computer to connect to the Windows Terminal Service.

To configure the external computer (e.g., in Windows 7 operating systems):

▶ Press the Windows Start button and select **Control Panel** on the Start menu.
▶ Select **System and Security**
▶ Select **System**
▶ Select **Remote settings**
▶ Under **Remote Assistance**, enable **Allow Remote Assistance connections to this computer**
▶ Under **Remote Desktop**, select **Allow connections from computers running any version of Remote Desktop**
▶ Press **OK** to confirm your settings

Configuring the control

To configure the control:

▶ Press the **DIADUR** key to open the **HEROS menu**
▶ Select **Remote Desktop Manager**
> The control opens the **Selection of server operating sys.** pop-up window.
▶ Select the desired operating system
  ◾ Win XP
  ◾ Win 7
  ◾ Win 8.X
  ◾ Win 10
  ◾ Other Windows version
▶ Press **OK**
> The control opens the **Edit the connection** pop-up window.
▶ Define the connection settings

Setting	Meaning	Input
Connection name	Name of the connection in **Remote Desktop Manager** ℹ️ You can use the following characters in the name of the connection: A B C D E F G H I J K L M N O P Q R S T U V W X Y Z a b c d e f g h i j k l m n o p q r s t u v w x y z 0 1 2 3 4 5 6 7 8 9 _ When you edit an existing connection, the control will automatically delete all impermissible characters from the name.	Required
Restarting after end of connection	Behavior after disconnection: ▪ Always restart ▪ Never restart ▪ Always after an error ▪ Ask after an error	Required
Automatic starting upon login	Connection is automatically established during control startup	Required
Add to favorites	Connection icon in the task bar: ▶ Single click with the left mouse button > The control switches to the desktop of the connection. ▶ Single click with the right mouse button > The control displays the connection menu.	Required
Move to the following workspace	Number of the desktop for the connection, desktop 0 being reserved for the NC software Default setting: Third desktop Select desktop 1 as the workspace or desktop for the connection	Required
Release USB mass memory	Permit access to connected USB mass memory devices	Required
Private connection	Connection can only be seen and used by its creator	Required
Computer	Host name or IP address of the external computer HEIDENHAIN recommends the following setting for the IPC(6641): **IPC6641.machine.net** The host name **IPC6641** must be assigned to the IPC in the Windows operating system for this setting. ℹ️ The **.machine.net** code is very important in this context. When **.machine.net** is specified, the control automatically searches the Ethernet interface **X116**, and not the interface **X26**—this reduces the time needed for access.	Required
User name	Name of the user	Required
Password	User password	Required
Windows domain	Domain of the external computer	Optional
Full-screen mode or **User-defined window size**	Size of the connection window	Required

Setting	Meaning	Input
Multimedia enhancements	Enables hardware acceleration when playing videos Certain formats, e.g. MP4 files, require the paid Fluendo Codec Pack ⚙ Any additional software will be installed by your machine tool builder.	Optional
Touchscreen input	Enables operation of multitouch systems and applications	Optional
Encryption	Sets the encryption that is appropriate for the selected Windows system ℹ If you activate the **Encryption** function, you need to remove the **-sec-tls -sec-nla** entries from the **Additional options** input field. If problems occur, try to set up a connection with the encryption function deactivated. The Windows log files are required for analysis.	Required
Color depth	Setting for the display of the external system on the control	Required
Locally effective keys	Shortcuts for automatically switching to the next active connection and workspace or desktop Default setting: ■ **Super_R** corresponds to the right DIADUR key and switches between the active connections ■ **F12** switches between the workspaces or desktops ℹ With the touchscreen of the MANUALplus 620, you can select the workspace or desktop via the touch keyboard. It is possible to adjust the default settings or to make additional entries	Required
Max. connection time (seconds)	Waiting time for connection If this timeout is exceeded, the connection could not be established	Required
Additional options	Available only to authorized specialists Additional command lines with transfer parameters ℹ If you activate the **Encryption** function, you need to remove the **-sec-tls -sec-nla** entries from the **Additional options** input field.	Required
Pass USB devices through	Passing the USB devices (e.g. 3-D mouse for operating CAD programs) connected to the control through to the Windows computer This requires installation of the Eltima EveUSB software on the Windows computer. ℹ All USB devices that have been passed through are not available on the control while the connection to the Windows computer is active.	Optional

For integrating the IPC 6641, HEIDENHAIN recommends the use of a RemoteFX connection.

When using a RemoteFX connection, the screen of the external computer is not mirrored, as with VNC. Instead, a separate desktop is opened. The desktop that is active on the external computer when establishing the connection is then locked and the user is logged off. This prevents two users from accessing the control simultaneously.

Configuring the connection – VNC

Configuring an external computer

ⓘ You need an additional VNC server for your external computer when trying to connect to VNC.

Install and configure the VNC server, e.g. the TightVNC server, before configuring the control.

▶ Press the **DIADUR** key to open the **HEROS menu**
▶ Select **Remote Desktop Manager**
> The control opens the **Remote Desktop Manager**.
▶ Press **New connection**
▶ Press **VNC**
> The control opens the **Edit the connection** pop-up window.
▶ Define the connection settings

Setting	Meaning	Input
Connection name:	Name of the connection in **Remote Desktop Manager**	Required
	ⓘ You can use the following characters in the name of the connection: A B C D E F G H I J K L M N O P Q R S T U V W X Y Z a b c d e f g h i j k l m n o p q r s t u v w x y z 0 1 2 3 4 5 6 7 8 9 _ When you edit an existing connection, the control will automatically delete all impermissible characters from the name.	
Restarting after end of connection:	Behavior after disconnection: ▪ Always restart ▪ Never restart ▪ Always after an error ▪ Ask after an error	Required
Automatic starting upon login	Connection is automatically established during control startup	Required
Add to favorites	Connection icon in the task bar: ▶ Single click with the left mouse button > The control switches to the desktop of the connection. ▶ Single click with the right mouse button > The control displays the connection menu.	Required
Move to the following workspace	Number of the desktop for the connection, desktops 0 and 1 being reserved for the NC software Default setting: Third desktop	Required
Release USB mass memory	Permit access to connected USB mass memory devices	Required
Private connection	Connection can only be seen and used by its creator	Required
Calculator	Host name or IP address of the external computer. In the recommended configuration of the IPC 6641, the IP address 192.168.254.3 is used.	Required
User name:	Name of the user to be logged on	Required

Setting	Meaning	Input
Password	Password for connecting to the VNC server	Required
Full-screen mode or **User-defined window size:**	Size of the connection window	Required
Permit further connections (share)	Additionally grant other VNC connections access to the VNC server	Required
View only	In display mode, the external computer cannot be operated	Required
Entries in the **Advanced options** area	Available only to authorized specialists	Optional

With VNC, the screen of the external computer is mirrored directly. The active desktop at the external computer is not locked automatically.

With a VNC connection, it is also possible to completely shut down the external computer via the Windows menu. As the computer cannot be rebooted via a connection, it must actually be switched off and on again.

Shutting down or rebooting an external computer

+---+
| ***NOTICE*** |
+---+
| **Caution: Data may be lost!** |
| |
| If you do not shut down external computers |
| properly, data may be irreversibly damaged or |
| deleted. |
| |
| ▶ Configure the automatic shutdown of the Windows |
| computer |
+---+

To configure the control:
▶ Press the **DIADUR** key to open the **HEROS menu**
▶ Select **Remote Desktop Manager**
> The control opens **Remote Desktop Manager**.
▶ Press **New connection**
▶ Press **Switch-off/restart of a computer**
> The control opens the **Edit the connection** pop-up window.
▶ Define the connection settings

Setting	Meaning	Input
Connection name:	Name of the connection in **Remote Desktop Manager**	Required
	ⓘ You can use the following characters in the name of the connection: A B C D E F G H I J K L M N O P Q R S T U V W X Y Z a b c d e f g h i j k l m n o p q r s t u v w x y z 0 1 2 3 4 5 6 7 8 9 _ When you edit an existing connection, the control will automatically delete all impermissible characters from the name.	
Restarting after end of connection:	Not necessary with this connection	–

Setting	Meaning	Input
Automatic starting upon login	Not necessary with this connection	–
Add to favorites	Connection icon in the taskbar: ▶ Single click with the left mouse button > The control switches to the desktop of the connection. ▶ Single click with the right mouse button > The control displays the connection menu.	Required
Move to the following workspace	Not active with this connection	–
Release USB mass memory	Not advisable with this connection	–
Private connection	Connection can be seen and used only by its creator	Required
Calculator	Host name or IP address of the external computer. In the recommended configuration of the IPC 6641, the IP address 192.168.254.3 is used.	Required
User name	User name to be used for establishing the connection	Required
Password	Password for connecting to the VNC server	Required
Windows domain:	Domain of the target computer, if required	Optional
Max. waiting time (seconds):	A shutdown of the control causes the Windows computer to shut down as well. Before the control displays the message **Now you can switch off.**, it waits for **<Timeout>** seconds. While waiting, the control checks whether the Windows computer is still accessible (port 445). If the Windows computer is switched off before the **<Timeout>** seconds have expired, the control will wait no longer.	Required
Additional waiting time:	Waiting time after the Windows computer has stopped being accessible. Windows applications may delay the shutdown of the computer after port 445 has been closed.	Required
Force	Close all programs on the Windows computer, even if dialogs are still open. If **Force** is not selected, Windows waits up to 20 seconds. This delays the shutdown process or the Windows computer is switched off before Windows has shut down.	Required
Restart	Reboot the Windows computer	Required
Run during restart	Reboot the Windows computer when the control reboots. Effective only if the control is rebooted using the shutdown icon at the bottom right in the taskbar or if it is rebooted as a result of a change in the system settings (e.g. network settings).	Required
Run during switch-off	The Windows computer is switched off when the control shuts down (no reboot). This is the standard scenario. Even the **END** key will then no longer trigger a reboot.	Required
Entries in the **Advanced options** area	Available only to authorized specialists	Optional

Starting and stopping the connection

Once a connection has been configured, this connection is shown as an icon in the **Remote Desktop Manager** window. When you highlight a connection, the **Start the connection** and **Terminate the connection** menu items become selectable.

If the desktop of the external connection or the external computer is active, all inputs from the mouse and the alphabetic keyboard are transmitted there.

When the HEROS 5 operating system is shut down, the control automatically terminates all connections. Please note, however, that only the connection is terminated, whereas the external computer or the external system is not shut down automatically.

Further information: "Shutting down or rebooting an external computer", Page 690

To switch between the third desktop and the control interface:

- Use the right DIADUR key on the alphabetic keyboard
- Use the taskbar
- Use the operating mode key

Exporting and importing connections

The **Export the connections** and **Import connections** functions allow you to save and restore the connections of **Remote Desktop Manager**.

ℹ️ To create and edit public connections when user administration is active, you need the HEROS.SetShares role. Users without this role are able to start and terminate public connections, but can only import, create, and edit private connections.

To export a connection, proceed as follows:

▶ Press the **DIADUR** key to open the **HEROS menu**
▶ Select **Remote Desktop Manager**
> The control opens the **Remote Desktop Manager**.
▶ Select the desired connection
▶ Select the right arrow symbol in the menu bar
> The control displays a selection menu.
▶ Select **Export the connections**
> The control opens a pop-up window.
▶ Define the name of the saved file
▶ Select the target file
▶ Select **Save**
> The control saves the connection data under the name defined in the pop-up window

To Import a connection, proceed as follows:

▶ Open the **Remote Desktop Manager**
▶ Select the right arrow symbol in the menu bar
> The control displays a selection menu.
▶ Select **Import connections**
> The control opens a pop-up window.
▶ Select file
▶ Select **Open**
> The control creates the connection under the name that was defined originally in the **Remote Desktop Manager**.

Private connections

Every user can create private connections by means of the user administration. A private connection can only be seen and used by the user who created it.

> ⓘ
> - If you create private connections before user administration was activated, these connections are no longer available when user administration is active.
>   Change private connections to public connections or export the connections before activating user administration.
> - To create and edit public connections, you need the HEROS.SetShares right. Users without this right are able to start and terminate public connections, but can only import, create, and edit private connections.
>   **Further information:** "Definition of roles", Page 751

To create a private connection, proceed as follows:

▶ Press the **DIADUR** key to open the **HEROS menu**

▶ Select **Remote Desktop Manager**

> The control opens the **Remote Desktop Manager**.

▶ Select **New connection**

▶ Select the desired connection (e.g., **Switch-off/restart of a computer**)

> The control opens the **Edit the connection** pop-up window.

▶ Define the connection settings

▶ Select **Private connection**

▶ Press **Ok**

> The control creates a private connection.

The control indicates private connections with a symbol:

Symbol	Meaning
	Public connection
	Private connection

You can save the connections individually with the **Export the connections** function.

Further information: "Exporting and importing connections", Page 693

When user administration is active, the control saves the private connections in the user's **HOME:** directory. If you create a backup with the **NC/PLC Backup** HEROS function, the control also saves the private connections. You can choose whether the control is to save the **HOME:** directory for the current user or for all users.

12.2 Additional tools for ITCs

The following additional tools allow you to apply various settings for the touch screens on connected ITCs.

ITCs are industrial PCs without their own memory media, and therefore they do not have their own operating system. This feature is what makes ITCs different from IPCs.

ITCs are frequently used with large machinery, e.g. as a clone of the actual control system.

> ⚙ Refer to your machine manual.
> The machine manufacturer defines and configures the display and function of the connected ITCs and IPCs.

Additional tool	Application
ITC Calibration	4-point calibration
ITC Gestures	Configuration of gesture control
ITC touchscreen configuration	Selection of touch sensitivity

> ℹ The additional tools for the ITCs are only provided by the control in the taskbar with connected ITCs.

Using the additional tool **ITC Calibration,** you align the position for the mouse cursor displayed with the actual movement position of your finger.

Calibration using the additional **ITC Calibration** tool is recommended in the following cases:

- After replacing the touchscreen
- When changing the touch screen position (parallel axis error due to amended viewing angle)

For calibration:

- ▶ Start the tool in control using the task bar
- > The ITC opens the calibration screen with four touch points in the corners of the screen
- ▶ Touch the four touch points shown one after the other
- > The ITC closes the calibration screen once calibration has been successfully completed

Using the additional **ITC Gestures** tool, the machine manufacturer configures the gesture control on the touch screen.

> ⚙ Refer to your machine manual.
> This function may only be used with the permission of your machine manufacturer.

Using the additional **ITC Touchscreen Configuration** tool, you can select the touch sensitivity of the touch screen.

The ITC gives you the following options:

- **Normal Sensitivity (Cfg 0)**
- **High Sensitivity (Cfg 1)**
- **Low Sensitivity (Cfg 2)**

Use the **Normal Sensitivity (Cfg 0)** setting as standard. If you find it difficult to operate the equipment while wearing gloves in this setting, select the **High Sensitivity (Cfg 1)** setting.

> **ℹ**   If the ITC touch screen is not splash-proof, select the **Low Sensitivity (Cfg 2)** setting. This stops the ITC interpreting drops of water as touches.

For configuration:

▶ Start the tool in control using the task bar
> The ITC opens a pop-up window with three options
▶ Select Touch Sensitivity
▶ Press the **OK** button
> The ITC closes the pop-up window

12.3 Window Manager

Refer to your machine manual.

The machine manufacturer determines the scope of function and behavior of the window manager.

The control features the Xfce window manager. Xfce is a standard application for UNIX-based operating systems, and is used to manage graphical user interfaces. The following functions are possible with the window manager:

- Display a 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

The control shows a star in the upper left of the screen if an application of the window manager or the window manager itself has caused an error. In this case, switch to the window manager and correct the problem. If required, refer to your machine manual.

Overview of taskbar

In the task bar, you can choose different workspaces by mouse click.

The control provides the following workspaces:

- Workspace 1: Interface of the control
- Workspace 2: Applications of the machine manufacturer (optionally available)
- Workspace 3: Applications of the machine tool builder (optionally available)
- Workspace 4: Display and remote control of external computer units (option 133) or applications of the machine tool builder (optionally available)

You can also select other applications from the taskbar which you have started in addition to the control software, e.g. **TNCguide**.

All open applications to the right of the green HEIDENHAIN icon can be moved between the workspaces any way you desire by pressing and holding the left mouse button.

Click the green HEIDENHAIN icon to open a menu in which you can get information, make settings or start applications.

Area	Function
Header	■ User name **Further information:** "Current User", Page 766 ■ 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:** "Switching or logging out users", Page 763
Navigation	■ Favorites ■ Recently used
Diagnostic	■ **GSmartControl**: Available only to authorized specialists ■ **HeLogging**: Define settings for internal diagnostic files ■ **perf2**: Check processor load and process load ■ **Portscan**: Test active connections **Further information:** "Portscan", Page 701 ■ **Portscan OEM**: Available only to authorized specialists ■ **Terminal**: Enter and execute console commands ■ **TNCdiag**: Evaluates status and diagnostic information of HEIDENHAIN components (particularly drives) and presents it graphically ⚙ If you want to use **TNCdiag**, contact your machine manufacturer. ■ **TNCscope**: Software for data recording

Area	Function
Settings	■ **Adjust screen brightness**: Adjust screen brightness
	■ **Screensaver**: Set the screensaver
	Further information: "Screensaver with lockout", Page 763
	■ **Current User**
	Further information: "Current User", Page 766
	■ **Date/Time**: Set date and time
	■ **Firewall**: Configure the firewall
	Further information: "Firewall", Page 708
	■ **Language/Keyboards**: Select system dialog language and keyboard version—the control overwrites the user interface language setting when starting based on the language setting of machine parameter **CfgDisplayLanguage** (no. 101300)
	■ **Network**: Define network settings
	Further information: "Ethernet interface ", Page 719
	■ **OEM Function Users**: Edit machine manufacturer function user
	Further information: "HEIDENHAIN function users", Page 750
	■ **PKI Admin**: Manage control certificates
	■ **Printer**: Configure and manage printer
	Further information: "Printers", Page 701
	■ **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).
	■ **SELinux**: Define safety software for Linux-based operating systems
	Further information: "SELinux security software", Page 732
	■ **Shares**: Connect and manage external network drives
	Further information: "Settings for network drives", Page 729
	■ **UserAdmin**: Configure user administration
	Further information: "Configuring the user administration", Page 734
	■ **VNC**: Define the setting for external software accessing the control, e.g., for maintenance procedures (Virtual Network Computing)
	Further information: "VNC", Page 703
	■ **WindowManagerConfig**: Available only to authorized specialists
Info	■ **About HeROS**: Display information on the operating system of the control
	■ **About Xfce**: Display information on the Window manager

Area	Function
Tools	■ **Switch-off**: Shut down the control
	Further information: "Switching or logging out users", Page 763
	■ **Screenshot**: Create screenshot
	■ **File Manager**: Available only to authorized specialists
	■ **Diffuse Merge Tool**: Compare and merge text files
	■ **Document Viewer**: Display and print files (e.g., PDF files)
	■ **Geeqie**: Open, manage, and print graphics
	■ **Gnumeric**: Open, edit, and print tables
	■ **IDS Camera Manager**: Manage cameras connected to the control
	■ **Leafpad**: Open and edit text files
	■ **NC Control**: Start or stop the NC software independently of the operating system
	■ **NC/PLC Backup**: Create backup file
	Further information: "Backup and restore", Page 706
	■ **NC/PLC Restore**: Restore backup file
	Further information: "Backup and restore", Page 706
	■ **QupZilla**: Alternative web browser for touch operation
	■ **Real VNC Viewer**: Define the setting for external software accessing the control (e.g., for maintenance purposes like Virtual Network Computing)
	■ **Remote Desktop Manager** (option 133)
	Further information: "Remote Desktop Manager (option 133)", Page 684
	■ **Ristretto**: Open graphics
	■ **Secure Remote Access**: Start and stop remote maintenance
	Further information: "Secure Remote Access", Page 712
	■ **TNCguide**: Access the help system
	■ **TouchKeyboard**: Open keyboard for touch operation
	■ **Web Browser**: Start the web browser
	■ **Xarchiver**: Extract or compress directories
Search	Full-text search of individual functions

Portscan

The PortScan function allows the cyclic or manual searching for all open, incoming TCP and UDP listen ports on the system. All ports found are compared with whitelists. If the control finds a non-listed port it shows a corresponding pop-up window.

The **HEROS menu Diagnostic** menu contains the **Portscan** and **Portscan OEM** applications for this purpose. **Portscan OEM** is only executable after entering the machine manufacturer password.

The **Portscan** function searches for all open, incoming TCP and UDP ports on the system and compares them to four whitelists stored in the system:

- System-internal whitelists **/etc/sysconfig/portscan-whitelist.cfg** and **/mnt/sys/etc/sysconfig/portscan-whitelist.cfg**
- Whitelist for ports for machine manufacturer-specific functions, e.g. for Python applications, external applications: **/mnt/plc/etc/sysconfig/portscan-whitelist.cfg**
- Whitelist for ports with customer-specific functions: **/mnt/tnc/etc/sysconfig/portscan-whitelist.cfg**

For each entry, the whitelist specifies the type of port (TCP/UDP), the port number, the providing program, and optional comments. If the automatic port scan function is active, only ports listed in the whitelists can be open. Non-listed ports trigger a notification window.

The result of the scan is saved to a log file (LOG:/portscan/scanlog and LOG:/portscan/scanlogevil) and, if new ports are found that are not listed in one of the whitelists, these are displayed.

Manually starting Portscan

To manually start the Portscan:

▶ Open the task bar at the bottom edge of the screen
Further information: "Window Manager", Page 697
▶ Press the green HEIDENHAIN button to open the HEROS menu
▶ Select the **Diagnostic** menu item
▶ Select the **Portscan** menu item
> The control opens the **HEROS Portscan** pop-up window.
▶ Press the **Start** key

Cyclically starting Portscan

To automatically start the Portscan cyclically:

▶ Open the task bar at the bottom edge of the screen
▶ Press the green HEIDENHAIN button to open the **HEROS menu**
▶ Select the **Diagnostic** menu item
▶ Select the **Portscan** menu item
> The control opens the **HeRos Portscan** pop-up window.
▶ Press the **Automatic update on** key
▶ Set the time interval with the slider

Printers

The **Printer** function in the **HEROS menu** allows you to add and manage printers.

Displaying the printer settings

To open the printer settings:

▶ Open the taskbar at the bottom edge of the screen
▶ Press the green HEIDENHAIN button to open the **HEROS menu**
▶ Select the **Settings** menu item
▶ Select the **Printer** menu item
> The control opens the **Heros Printer Manager** dialog.

Soft key	Function	Meaning
CREATE	**Create**	Creates a printer
CHANGE	**CHANGE**	Adapts the properties of the selected printer
COPY	**COPY**	Creates a copy of the selected printer Initially, the copy has the same properties as the copied printer. This can be useful for printing both portrait and landscape formats on the same printer
DELETE	**DELETE**	Deletes the selected printer
UP	**UP**	Selects a printer
DOWN	**DOWN**	
STATUS	**STATUS**	Displays the status information of the selected printer
PRINT A TEST PAGE	**PRINT A TEST PAGE**	Prints a test page on the selected printer

ℹ The connected printer must be PostScript-enabled.

VNC

Use the **VNC** function to configure the behavior of the various VNC clients. This includes, for example, operation via soft keys, mouse, and the alphabetic keyboard.

The control provides the following options:

■ List of permitted clients (IP address or name)
■ Password for the connection
■ Additional server options
■ Additional settings for assigning the focus

⚙ Refer to your machine manual.

For multiple clients or operating units, the focus assignment procedure depends on the design and the operating situation of the machine.

This function must be adapted by your machine manufacturer.
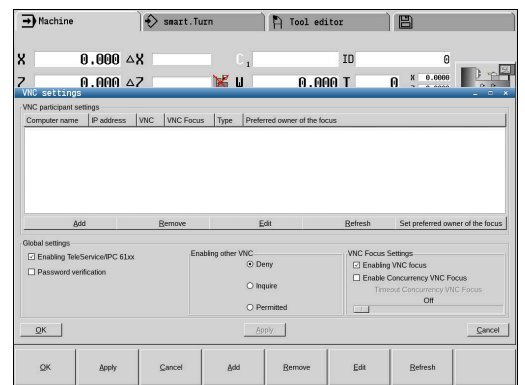
Opening VNC settings

To open the **VNC settings**:

To open the **VNC settings**:

▶ Open the task bar at the bottom edge of the screen

▶ Press the green HEIDENHAIN button to open the **HEROS menu**

▶ Select the **Settings** menu item

▶ Select the **VNC** menu item

> The control opens the **VNC settings** pop-up window.

The control provides the following options:

■ Add: Add new VNC viewer/client

■ Remove: Deletes the selected client. Only possible with manually entered clients.

■ Edit: Edit the configuration of the selected client

■ Update: Updates the display. Required with connection attempts during which the dialog is open.

VNC settings

Dialog	Option	Meaning
VNC participant settings	**Computer name**	IP address or computer name
	VNC	Connection of the client to the VNC viewer
	VNC Focus	The client participates in the focus assignment
	Type	■ Manual Manually entered client ■ Denied This client is not permitted to connect ■ Enable TeleService and IPC Client via TeleService connection ■ DHCP Other computer that obtains an IP address from this computer
Firewall warning		Warnings and information about if the VNC protocol has not been authorized for all VNC clients due to firewall settings on the control. **Further information:** "Firewall", Page 708.
Global settings	**Enable RemoteAccess and IPC**	Connection is always permitted
	Password verification	The client must enter a password for verification. If this option is active the password must be entered when the connection is established.

Dialog	Option	Meaning
Enabling other VNC	**Deny**	Access generally denied to all other VNC clients.
	Inquire	During connection attempts a corresponding dialog is opened.
	Permitted	Access is generally granted to all other VNC clients.
VNC Focus Settings	**Enabling VNC focus**	Enable focus assignment for this system. Otherwise there is no central focus assignment. In the default setting, the focus is actively reassigned by the owner of the focus by clicking the focus symbol. This means that the owner of the focus must first release the focus by clicking the focus symbol before any other client can retrieve the focus.
	Enable Concurrency VNC Focus	In the default setting, the focus is actively reassigned by the owner of the focus by clicking the focus symbol. This means that the owner of the focus must first release the focus by clicking the focus symbol before any other client can retrieve the focus. If concurrency focus is selected, any client can retrieve the focus at any time without having to wait for the current owner of the focus to release it.
	Timeout Concurrency VNC Focus	Time period within which the current owner of the focus can object to the focus being withdrawn or can prevent the reassignment of the focus. If a client requests the focus, a dialog in which the reassignment of focus can be refused appears on all clients' screens.
Focus symbol	(symbol)	Current status of VNC focus on the respective client: Focus is owned by other client. Mouse and alphabetic keyboard are disabled.
	(symbol)	Current status of VNC focus on respective client: Focus is owned by current client. Entries can be made.
	(symbol)	Current status of VNC focus on the respective client: Request by the owner of the focus to give the focus to another client. Mouse and alphabetic keyboard are disabled until the focus is assigned unambiguously.

If **Enable Concurrency VNC Focus** is selected, a pop-up window appears. This dialog allows you to deny the focus change to the requesting client. If you fail to do so, the focus changes to the requesting client after the set time limit.

> Activate the **Enabling VNC focus** check box only in connection with HEIDENHAIN devices provided especially for this purpose (e.g., ITC industrial computers).

Backup and restore

With the **NC/PLC Backup** and **NC/PLC Restore** functions, you can back up and restore individual folders or the entire **TNC:** drive. You can save the backup files locally, on a network drive, or to USB storage devices.

The backup program generates a ***. tncbck** file that can also be processed by the PC tool TNCbackup (part of TNCremo). The Restore program can restore these files as well as those from existing TNCbackup programs. If you select a *. tncbck file in the File Manager of the control, the **NC/PLC Restore** program will be launched automatically.

Backup and restore is subdivided into several steps. Navigate between these steps with the **FORWARD** and **BACK** soft keys. Specific actions for steps are selectively displayed as soft keys.

Open NC/PLC Backup or NC/PLC Restore

To open the function:

▶ Open the task bar at the bottom edge of the screen
▶ Press the green HEIDENHAIN button to open the **HEROS menu**
▶ Select the **Tools** menu item
▶ Open the **NC/PLC Backup** or **NC/PLC Restore** menu item
> The control opens the pop-up window.

Backing up data

To back-up the control data:

▶ Select **NC/PLC Backup**
▶ Select
  type
  ■ Back up the **TNC:** drive
  ■ Back up the directory tree: Select the directory for backup in the file management
  ■ Backup machine configuration (for machine tool builders only)
  ■ Complete backup (for machine tool builders only)
  ■ Comment: Freely configurable comment for the backup
▶ Select the next step with the **FORWARD** soft key
▶ Stop the control if required with the **STOP NC SOFTWARE** soft key
▶ Define the exclusion rules
  ■ User preset rules
  ■ Write own rules to the table
▶ Select the next step with the **FORWARD** soft key
> The control generates a list of files for backing up.
▶ Check the list. Deselect files if necessary.
▶ Select the next step with the **FORWARD** soft key
▶ Enter the name of the backup file
▶ Select the storage path
▶ Select the next step with the **FORWARD** soft key
> The control generates the backup file.
▶ Confirm with the **OK** soft key
> The control concludes the backup process and restarts the NC software.

Restoring data

> | ***NOTICE*** |
> | :---: |
> | **Caution: Data may be lost!** |
> | When you restore data (Restore function), any existing data will be overwritten without a confirmation prompt. Existing data is not automatically backed up by the control before running the restore process. Power failures or other problems can interfere with the data restore process. As a consequence, data may be irreversibly damaged or deleted. |
> | ▶ Before starting the data restore process, make a backup of the existing data |

To restore data:

▶ Select **NC/PLC Restore**

▶ Select the archive to be restored

▶ Select the next step with the **FORWARD** soft key

> The control generates a list of files for restoring.

▶ Check the list. Deselect files if necessary.

▶ Select the next step with the **FORWARD** soft key

▶ Stop the control if required with the **STOP NC SOFTWARE** soft key

▶ Extract the archive

> The control restores the files.

▶ Confirm with the **OK** soft key

> The control restarts the NC software.

12.4 Firewall

Application

You have the possibility to set up a firewall for the primary network interface of the control and a sandbox, if needed. You can configure it so that incoming network traffic is blocked and/or a message is displayed, depending on the sender and the service.

The firewall cannot be started for the secondary network interface of the control (i.e., the machine network).

If you activate the firewall, the control displays a corresponding icon at the bottom right in the taskbar. The symbol changes depending on the safety level that the firewall was activated with, and informs about the level of the safety settings:

Icon	Meaning
	No firewall protection yet although it has been activated according to the configuration. This can happen, for example, if a dynamic IP address is used in the network interface configuration, but the DHCP server has not yet assigned such an address.
	Firewall active with medium security level
	Firewall is activated with a high security level All services except for SSH are blocked.

> **i** Have your network specialist check and, if necessary, change the standard settings.

Configuring the firewall

▶ Open the task bar at the bottom edge of the screen
▶ Set the **active** option for switching on the firewall
▶ Press the **Set standard values** button
▶ Confirm the changes with the **Apply** function
▶ Exit the dialog with the **OK** function

Firewall settings

Setting	Meaning
Active	Switch the firewall on or off
Interface	Select the interface
	Selection of the **eth0** interface usually corresponds to X26 of the MC main computer. **eth1** corresponds to X116.
	You can check the interfaces in the network settings on the Interfaces tab. For main computer units with two Ethernet interfaces, the DHCP server for the machine network is active for the second, non-primary, interface by default. With this setting, the firewall cannot be activated for **eth1** because the firewall and DHCP server mutually exclude each other.
Report other inhibited packets	Activate the firewall with a high security level
	All services except for SSH are blocked.
Inhibit ICMP echo answer	If this option is set, the control no longer responds to a PING request

Setting	Meaning
Service	This column contains the short names of the services that are configured with this dialog. For the configuration, it is not important here whether the services themselves have been started. ■ **DNC** designates the service the DNC server provides via the RPC protocol for external applications that were developed with the RemoTools SDK (port 19003) > 📖 For more detailed information, consult the RemoTools SDK manual. ■ **LDAPS** includes the server on which the user data and the user administration configuration are saved. ■ **LSV2** includes the functionality for **TNCremo**, Teleservice, and other HEIDENHAIN PC Tools (port 19000) > 🛈 When user administration is active, you can set up only secure network connections via SSH. The control automatically disables the LSV2 connections via the serial interfaces (COM1 and COM2) and the network connections without user authentication. > If user administration is inactive, the control also automatically blocks insecure 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 insecure connections. These machine parameters are contained in the **CfgDncAllowUnsecur** (135400) data object. ■ **SMB** only refers to incoming SMB connections (i.e., if a Windows share is created on the NC). Outgoing SMB connections (i.e., if a Windows share is connected to the NC) are not affected by this setting. ■ **SSH** stands for the Secure Shell protocol (port 22). As of HEROS 504, LSV2 can be executed securely via this SSH protocol while user administration is active. **Further information:** "User authentication from external applications", Page 756 ■ **VNC** protocol means access to the screen contents. If you block this service, the screen content can no longer be accessed, not even with the Teleservice programs from HEIDENHAIN (e.g., to create a **Screenshot**) In this case, the HEROS VNC configuration dialog shows a warning that **VNC** is disabled in the firewall.
Method	Configure accessibility Under **Method**, you can configure whether the service is accessible: ■ **Prohibit all**, cannot be accessed by anyone ■ **Permit all**, can be accessed by everyone ■ **Permit some**, can be accessed only by individual clients If you set **Permit some**, you must also specify the computer that you wish to grant access to the respective service. If you do not specify anything under **Computer**, the setting **Prohibit all** will automatically become active when the configuration is saved.
Log	If **Log** is active, the control outputs a **red** message when a network packet for this service has been blocked. The control shows a blue message when a network packet for this service has been accepted.

Setting	Meaning
Calculator	If you have selected the **Permit some** setting under **Method**, specify the computers here.
	The computers can be entered with their IP addresses or with a host name. Separate multiple computers by commas. If you use a host name, the system checks upon closing or saving the dialog whether this host name can be translated into an IP address. If this is not the case, an error message is displayed and the dialog will not terminate.
	If you specify a valid host name, this host name is translated into an IP address each time the control is started. If a computer that was entered with its name changes its IP address, you may have to restart the control or formally change the firewall configuration. In this case, the control uses the new IP address for a host name in the firewall.
Advanced options	These settings are only intended for your network specialists
Set standard values	Reset the settings to the default values recommended by HEIDENHAIN

12.5 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: "VNC", Page 703

Requirements

■ Existing Internet connection

Further information: "Editing network connection window", Page 725

■ 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: "VNC", Page 703

■ 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 an 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
Connect	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.
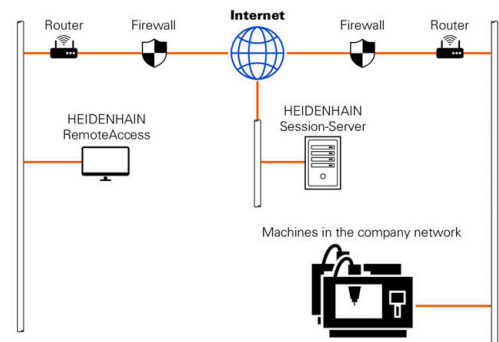
12.6 Software for data transfer

Application

HEIDENHAIN offers the TNCremo software for connecting a Windows PC to a HEIDENHAIN control in order to transfer data.

Prerequisites

System requirements for TNCremo:

- PC operating system:
    - Windows 8
    - Windows 10
- PC RAM: 2 GB
- Free PC hard-disk space: 15 MB
- A network connection to the control

Functionality

The TNCremo data transfer software provides the following areas:
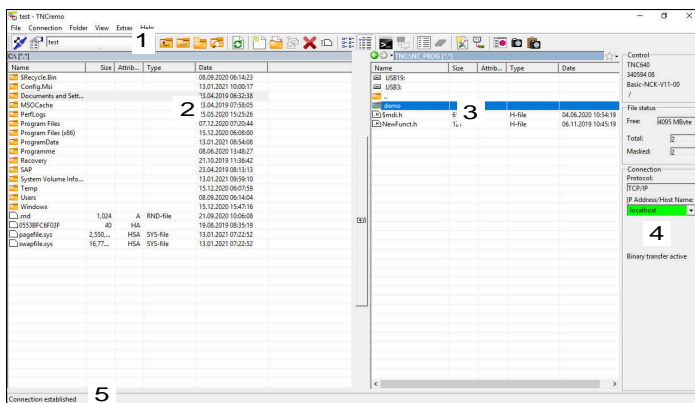
1   Toolbar

    This area provides the most important TNCremo functions.

2   File list of 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.

Installing TNCremo

To install TNCremo on a PC:

▶ Start the SETUP.EXE installation program from the Explorer
▶ Follow the installer instructions

Starting TNCremo

To start TNCremo in Microsoft Windows 10:

▶ Press the Windows key
▶ Select the HEIDENHAIN folder
▶ Select TNCremo

  or

▶ Double-click the TNCremo desktop icon

Configuring a connection

Before connecting to the control, you need to configure a connection.

To configure a connection:

▶ Press the **Configure** button
> TNCremo opens the **Connection configuration** window.
▶ Select the **Management** tab
▶ Select **New...**
> TNCremo opens the **New configuration** window.
▶ Enter the connection name
▶ Press **OK**
> TNCremo automatically opens the **Connection** tab.
▶ Select the **Type of connection**

ℹ If you use the default type of connection, a network connection (TCP/IP) is set up over the Ethernet interface.

▶ Select the **Settings** tab
▶ Enter the **IP address/Host name** of the control
▶ Select **OK**
> TNCremo saves the configuration.
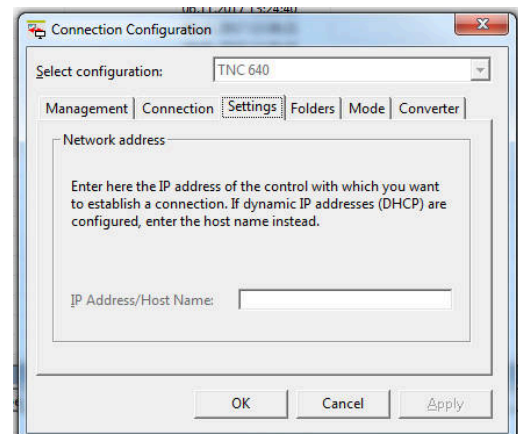
Establishing a connection to the control

Once you have configured a connection, you can connect your PC to the control.

To set up a connection to the control:

▶ Select the configured connection in the selection menu

> ▶ Select the **Set up connection** icon
> > The connection to the control will be established.
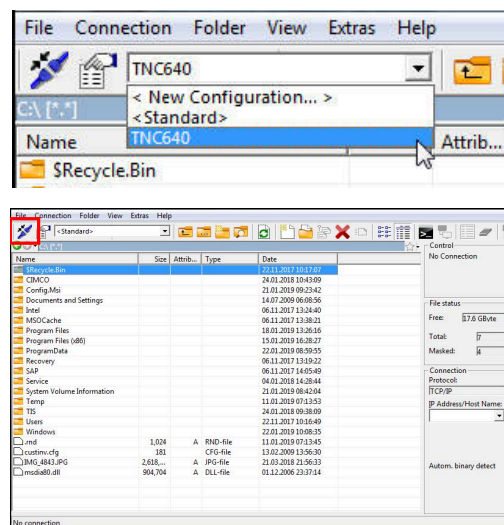
Switching the drive

You can switch the PC drive or the control drive that is shown in TNCremo.

To switch the shown drive:

▶ Select the PC's file list or the control's file list

> ▶ Select the **Change folder/drive** icon
> > TNCremo opens a pop-up window.
> ▶ Select the desired drive in the selection menu
> ▶ Select **OK**
> > TNCremo displays the selected drive.

Notes

- When user administration is active, you can set up only secure network connections via SSH. The control automatically disables the LSV2 connections via the serial interfaces (COM1 and COM2) and the network connections without user authentication.

  If user administration is inactive, the control also automatically blocks insecure 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 insecure connections. These machine parameters are contained in the **CfgDncAllowUnsecur** (135400) data object.

- You can download the current version of the TNCremo software from the **HEIDENHAIN website**.

12.7 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 733
- Principle of the SSH connection
  Further information: "User authentication from external applications", Page 756
- Firewall settings
  Further information: "Firewall", Page 708

Requirements

- PC software TNCremo with version 3.3 or higher is installed
  Further information: "Software for data transfer", Page 714
- **SSH** service is permitted in the firewall of the control
  Further information: "Firewall", Page 708

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 by using 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.

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

> ℹ️ 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**.

Notes

- When user administration is active, you can set up only secure network connections via SSH. The control automatically disables the LSV2 connections via the serial interfaces (COM1 and COM2) and the network connections without user authentication. If user administration is inactive, the control also automatically blocks insecure 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 insecure connections. These machine parameters are contained in the **CfgDncAllowUnsecur** (135400) data object.

- 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.

12.8 Ethernet interface

Introduction

The control is provided with an Ethernet interface as a standard feature so that you can integrate it into a network as a client.

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.

> ℹ️
> - 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.

Connection possibilities

You can integrate the Ethernet interface of your control into your network over the **RJ45** connection, or directly connect it to a PC. The connection is metallically isolated from the control electronics.

> ℹ️
> The maximum cable length between control and a node depends on the quality grade of the cable, the sheathing and the type of network.
>
> If you connect the control directly with a PC you must use a crossover cable.
>
> Have a network specialist configure the control.
>
> Please note that the control performs an automatic restart if you change the IP address of the control.
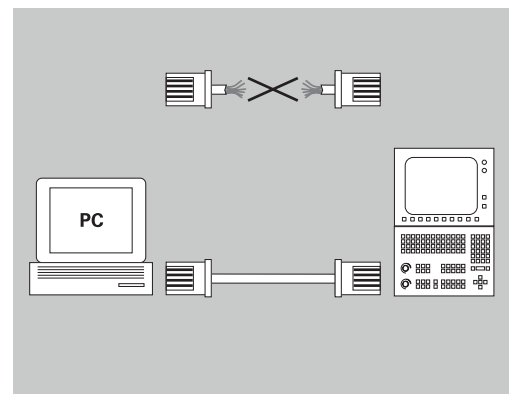
Ethernet connection icon

Icon	Meaning
🖥️	Ethernet connection
	The control displays the icon at the bottom right in the taskbar.
	Further information: "Overview of taskbar", Page 697
	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

Network settings window

In the **Network settings** window, you define the settings for the control's Ethernet interface.

🛈 Have a network specialist configure the control.

To open the **Network settings** window:
▶ Open the taskbar at the bottom edge of the screen
▶ Press the green HEIDENHAIN button
> The control opens the HEROS menu.
▶ Select **Settings**
▶ Select the **Network** menu item
> The control opens the **Network settings** window.

🛈 ▪ Preferably restart the control after making changes to the network settings.
▪ The HEROS operating system manages the **Network settings** window. You must restart the control in order to change the HEROS conversational language.

Status tab

The **Status** tab contains the following information and settings:

Domain	Information or Setting
Computer name	The control displays the name under which the control is shown in the company network. You can change the name.
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: "Interfaces tab", Page 722

Domain	Information or Setting
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.

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
Configura-tion 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 ■ Reconnect the Ethernet interface with **Reconnect** ■ Edit the selected profile **Further information:** "Editing network connection window", Page 725

> ℹ
> - 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.

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.

Ping/Routing tab

You can check the network connection on the **Ping/Routing** tab.

The **Ping/Routing** tab contains the following information and settings:

Domain	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.

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 or importing a network profile

To export a network profile:
- ▶ Opening 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** Use the **HEIDENHAIN presettings** button to import the default values of the network settings.

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.
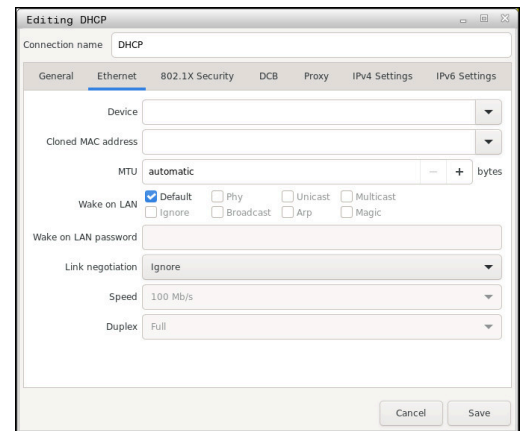
General tab

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

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

Setting	Meaning
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

802.1X Security tab

Currently no function

DCB tab

Currently no function

Proxy tab

Currently no function

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**

Setting	Meaning
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

IPv6 Settings tab

Currently no function

Settings for network drives

ℹ️ Have a network specialist configure the control.

To open the settings for network drives, proceed as follows:

▶ Open the task bar at the bottom edge of the screen
▶ Press the green HEIDENHAIN button
> The control opens the HEROS menu.
▶ Select the **Settings** menu item
▶ Select the **Shares** menu item
> The control displays the **Mount Setup** dialog.

In the **Network drive** area of the **Mount Setup** window, the control shows a list of all defined network drives and the status of each drive.

In the **Status Log** area, the control shows status information and error messages.
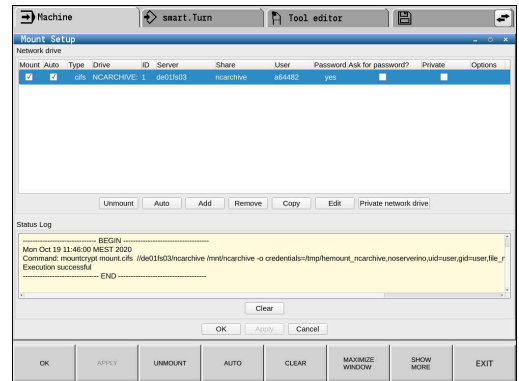
Button	Meaning
Mount	Connect a network drive
	The control ticks the check box in the **Mount** column when there is an active connection.
Unmount	Disconnect a network drive
Auto	Automatically connect the network drive whenever the control is switched on
	The control ticks the check box in the **Auto** column when there is an automatic connection.
Add	Define a new network drive
Unmount	Delete an existing network drive
Copy	Copy a network drive
Edit	Editing a network drive
Clear	Delete the contents of the **Status Log** area
Private network drive	User-specific network drive when user administration is active
	The control ticks the check box in the **Privat** column when there is a user-specific connection.

Adding a network drive

Prerequisites for connecting a network drive:

■ Connection with the network
■ The control must reach the server in the network
■ The credentials and the drive path are known

To add a network drive:

▶ Select **Add**
> The control opens the **Mount assistant** window.
▶ Define the settings on the individual tabs
▶ Select **Forward** after each tab
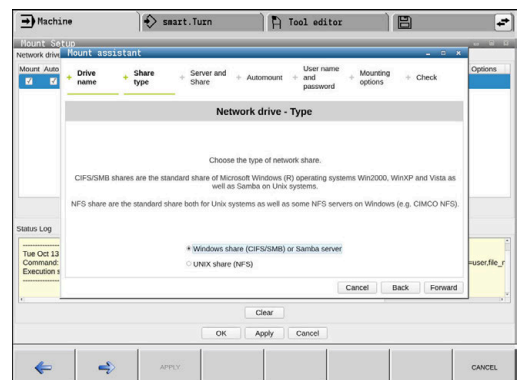
▶ On the **Check** tab, check the settings

▶ Select **Apply**

> The control connects the network drive.

The control guides you through the settings with the **Mount assistant**.

Tab	Setting
Drive name	■ **Drive name:** Network drive name shown in the file management of the control The names must be all uppercase letters, terminated by a colon (**:**). ■ **Private network drive** With user administration active, the connection is only visible to the user who created it. ℹ To create and edit public connections, you need the HEROS.SetShares right. Without this right, users are able to start and terminate public connections but can create and edit only private connections. **Further information:** "Definition of roles", Page 751
Share type	Transfer protocol ■ **Windows share (CIFS/SMB) or Samba server** ■ **UNIX share (NFS)**
Server and Share	■ **Server name:** Server name or IP address of network drive ■ **Share name:** Designation used to share the folder 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**

Tab	Setting
Mounting options	**Parameters for mount option "-o":** Auxiliary parameters for the connection
Check	The control displays a summary of the defined settings. You can check the settings and save them with **Apply**.

Enter options without a space, only separated by a comma

Options for NFS

Example	Meaning
rsize=8192	Packet size in bytes for data reception Input range 512 to 8192
wsize=4096	Packet size in bytes for data transmission Input range 512 to 8192
soft,timeo=3	Conditional Mount Time in tenths of a second after which the control will repeat the attempt
nfsvers=2	Protocol version

> **ⓘ** If you use the CIMCO NFS software, you must set this option. CIMCO NFS supports NFS only up to version 2.

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.

> **ⓘ** To avoid security gaps, prefer the current versions of the **SMB** and **NFS** protocols.
>
> Depending on the software status of the control, the network drive may require an older protocol version. In this case, you may change the protocol version with the **vers=** auxiliary parameter. Contact your network specialist.

12.9 SELinux security software

SELinux is an extension for Linux-based operating systems. **SELinux** is an additional security software in the sense of Mandatory Access Control (MAC) and protects the system against the execution of unauthorized processes or functions, and therefore against viruses and other malware.

MAC means that every action must be explicitly permitted, otherwise it will not be executed by the control. The software is intended as protection in addition to the normal access restriction in Linux. Certain processes and actions can only be executed if the standard functions and access control of **SELinux** permit it.

> ℹ️ The SELinux installation of the control has been prepared to permit running only programs installed with the HEIDENHAIN NC software. Other programs cannot be run with the standard installation.

The access control of **SELinux** under HEROS 5 is regulated as follows:

- The control executes only applications that are installed with the HEIDENHAIN NC software
- Files in connection with the security of the software (**SELinux** system files, HEROS 5 boot files, etc.), may only be modified by programs that have been selected explicitly.
- New files generated by other programs must never be executed
- USB data carriers cannot be deselected
- There are only two processes that are permitted to execute new files:
    - Starting a software update: A software update from HEIDENHAIN can replace or change system files
    - Starting the SELinux configuration: The configuration of **SELinux** is usually password-protected by your machine manufacturer; refer to the relevant machine manual.

> ℹ️ HEIDENHAIN recommends activating **SELinux** because it provides additional protection against attacks from outside.

12.10 User administration

Introduction

> ⚙ Refer to your machine manual.
>
> Some user administration areas are configured by the machine manufacturer.
>
> User administration is inactive in the control's factory default setting. This status is called **legacy mode**. In **legacy mode**, the control's behavior is identical to that of older software versions without user administration.
>
> It is not mandatory that you use user administration, but it is essential for the implementation of an IT security system.
>
> User administration supports you in the following fields of security, based on the requirements of the IEC 62443 series of standards:
>
> ■ Application security
>
> ■ Network security
>
> ■ Platform security
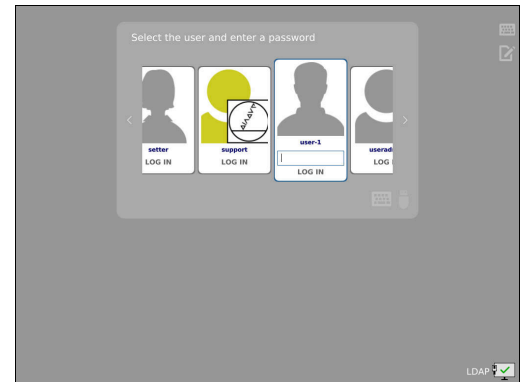
User administration enables you to define users with different access rights.

The following options are available for saving your user data:

■ **Local LDAP database**

■ Using 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

Further information: "Local LDAP database", Page 738

■ **LDAP on remote computer**

■ Importing an LDAP server configuration file

Further information: "LDAP on remote computer", Page 739

■ **Connection to Windows domain**

■ Integrating user administration on more than one control

■ Using different roles on different controls

Further information: "Logon to Windows domain", Page 740

> ℹ Parallel operation of Windows users and users from an LDAP database is possible.

Configuring the user administration

ℹ️   If you used **Remote Desktop Manager** to establish private connections before user administration was activated, these connections are no longer available after the activation of user administration.

Save your private connections before activating user administration.

Further information: "Remote Desktop Manager (option 133)", Page 684

User administration is inactive in the control's factory default setting. This status is called **Legacy-Mode**.

User administration needs to be configured before you can use it.

Call the user administration

To call user administration:

▶ Press the **DIADUR** key to open the **HEROS menu**

▶ Select the **Settings** menu item

▶ Select the **UserAdmin** menu item

> The control opens the **User administration** window.

ℹ️   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.

Activating user administration

To activate user administration:

▶ Call user administration

▶ Press the **User administration active** soft key

> The control shows the message **Password for user 'useradmin' missing**.
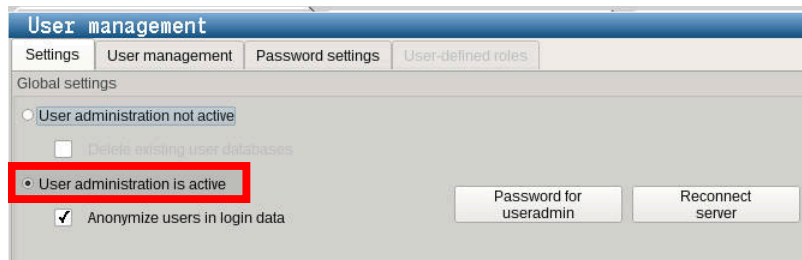
> ℹ The purpose of the **Anonymize users in log data** function is data privacy; this function is active by default. While this function is active, user data in all log files of the control will be anonymized.

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

---

Deactivating user administration

User administration can only be deactivated by the following function users:

- **useradmin**
- **OEM**
- **SYS**

To deactivate user administration:

- ▶ Log in as the appropriate function user
- ▶ Call user administration
- ▶ Select **User administration inactive**
- ▶ If necessary, select the **Delete existing user databases** check box to delete any configured users and user-specific directories.
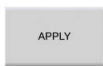
| APPLY | ▶ Press the **APPLY** soft key |

| EXIT | ▶ Press the **Exit** soft key |

- > The control opens the **System reboot required** window.
- ▶ Select **Yes**
- > The control triggers a restart.

Creating an useradmin function user

After user administration has been activated once, create the **useradmin** function user.

The **useradmin** user is comparable to the local administrator of a Windows system.

To create the **useradmin** user:

▶ Select **Password for useradmin**
> The control opens the **Password for user 'useradmin'** pop-up window.
▶ Set the password for the **useradmin** user
▶ Select **Set new password**
> The control shows the message **Settings and password for 'useradmin' were changed**.

ℹ For reasons of security, passwords should comply with the following criteria:
- Eight characters minimum
- Letters, numbers, and special characters
- Avoid using whole words or a sequence of characters (e.g., Anna or 123)

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.

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

ℹ The **useradmin** user automatically receives the HEROS.Admin role. This allows him to manage users in user administration, provided that he knows the LDAP database password. The **useradmin** user is a function user who has been pre-defined by HEIDENHAIN. You can neither add roles to function users, nor delete their roles.

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.

Setting up a database

To set up the database:

- ▶ Select the database for saving the user data
- ▶ Set up a database
- ▶ Press the **APPLY** soft key
- ▶ Press the **END** soft key
- > The control opens the **System reboot required** window.
- ▶ Press **Yes** to restart the system
- > The control restarts.
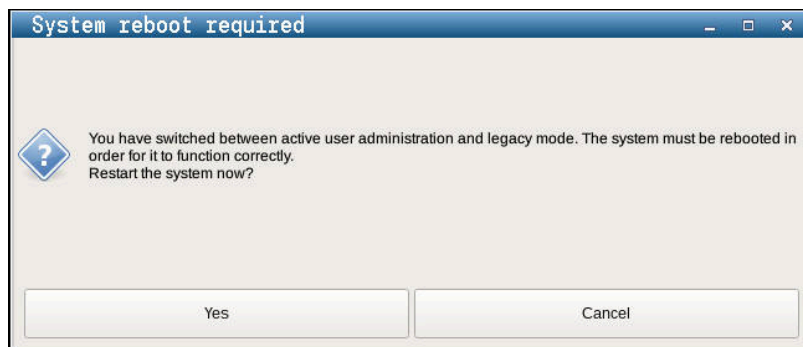
Local LDAP database

Before you can use the **Local LDAP database** function, the following requirements must be met:

- ■ User administration is active
- ■ The **useradmin** user has been configured

To set up a **Local LDAP database**:

- ▶ Call the user administration
- ▶ Select the **LDAP user database** function
- > The control enables the dimmed area for editing the LDAP user database.
- ▶ Select the **Local LDAP database** function
- ▶ Select the **Configuration** function
- > The control opens the **Configure local LDAP database** window.
- ▶ Enter the name of the **LDAP domain**
- ▶ Enter the password
- ▶ Repeat the password
- ▶ Press the **OK** soft key
- > The control closes the **Configure local LDAP database** window.
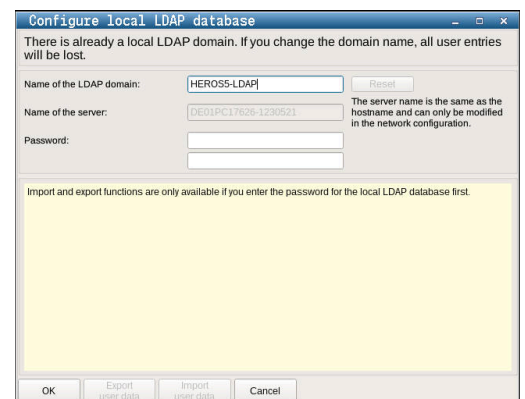
ℹ️ Before you can start editing the user administration, the control prompts you to enter the password of your local LDAP database.

Passwords must not be trivial and must be known only to the administrators.

Further information: "Creating further users", Page 744

ℹ️ If the host name or domain name of the control changes, you need to reconfigure the local LDAP databases.

LDAP on remote computer

Requirements

Before you can use the **LDAP on remote computer** function, the following requirements must be met:

- User administration is active
- The **useradmin** user has been configured
- An LDAP database has been set up in the company network
- A server configuration file of an existing LDAP database must be stored on the control or on a PC in the network
- The PC with the existing configuration file is running
- The PC with the existing configuration file is accessible in the network

Providing a server configuration file

To provide a server configuration file for an LDAP database:

- ▶ Call the user administration
- ▶ Select the **LDAP user database** function
- > The control enables the dimmed area for editing the LDAP user database.
- ▶ Select the **Local LDAP database** function
- ▶ Select the **Export server configuration** function
- > The control opens the **Export LDAP configuration file window.**
- ▶ Enter the name for the server configuration file into the name field
- ▶ Save the file to the desired folder
- > The server configuration file was exported successfully.

Using the LDAP database on a remote computer

To use the **LDAP on remote computer** function:

- ▶ Call the user administration
- ▶ Select the **LDAP user database** function
- > The control enables the dimmed area for editing the LDAP user database.
- ▶ Select the **LDAP on remote computer** function
- ▶ Select the **Import server configuration** function
- > The control opens the **Import LDAP configuration file window.**
- ▶ Select the existing configuration file
- ▶ Select **FILE**
- ▶ Press the **APPLY** soft key
- > The configuration file was imported.
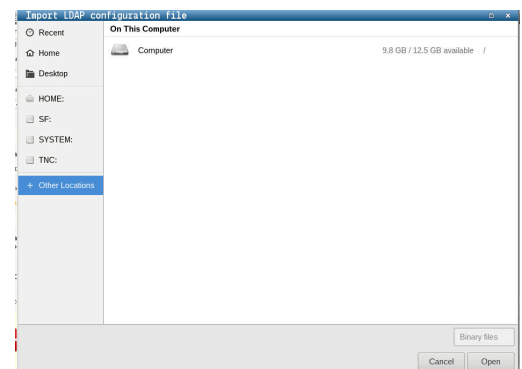
Logon to Windows domain

Prerequisites

Before you can use the **Connection to Windows domain** function, the following requirements must be met:

- User administration is active
- The **useradmin** user has been configured
- There is a Windows Active Domain Controller in the network
- The domain controller is accessible in the network
- Organizational unit for HEROS roles known
- For logon with computer account:
    - You have access to the password of the domain controller
    - You have access to the user interface of the domain controller or you are supported by an IT administrator
- For logon with function user:
    - User name of the function user
    - Password of the function user

Joining a Windows domain with a computer account

To join a Windows domain with a computer account:

▶ Call the user administration

▶ Select the **Connection to Windows domain** function

▶ Select the **Join Active Directory domain (with computer account)** check box

▶ Select the **Find domain** function

> **i** Use the **Configuration** function to define various settings of your connection:
>
> ■ Use the **Map SIDs to Unix UIDs** check box to select whether Windows SIDs are automatically mapped to Unix UIDs
>
> ■ Use the **Use LDAPs** check box to select LDAP or secure LDAPs. For LDAPs, define whether or not the secure connection verifies a certificate
>
> ■ Define a special group of Windows users to whom you want to restrict the connection to this control
>
> ■ Modify the organizational unit in which the HEROS role names are stored
>
> ■ 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)
>
> ■ Modify the separator within the HEROS role names

▶ Press the **APPLY** soft key

> The control opens the **Connect to domain** window.

> **i** The **Organizational unit for computer account:** function allows you to specify in which of the already existing organizational units you want to create the access, such as
>
> ■ ou=controls
>
> ■ cn=computers
>
> The values you enter must match the conditions of the domain. The terms are not exchangeable.

▶ Enter the user name of the domain controller

▶ Enter the password of the domain controller

> The control connects to the Windows domain found.

> The control checks whether all of the required roles have been created in the domain as groups.

	If not all of the required roles have been created in the domain as groups, the control issues a warning.
🛈	If the control issues a warning, proceed in one of the two following ways:
	▶ Press the **Add role definition** soft key
	■ Select the **Add** function
	You can enter the roles directly into the domain in this case.
	■ Press **Export**
	With this function, you can export the roles to a file in .ldif format.

> All of the required roles have been created in the domain as groups.

Joining a Windows domain with a function user

To join a Windows domain with a function user:

▶ Open the **User administration** window

▶ Select **Connection to Windows domain**

▶ Select the **Active Directory with function user** check box

▶ 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.

Creating groups

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:
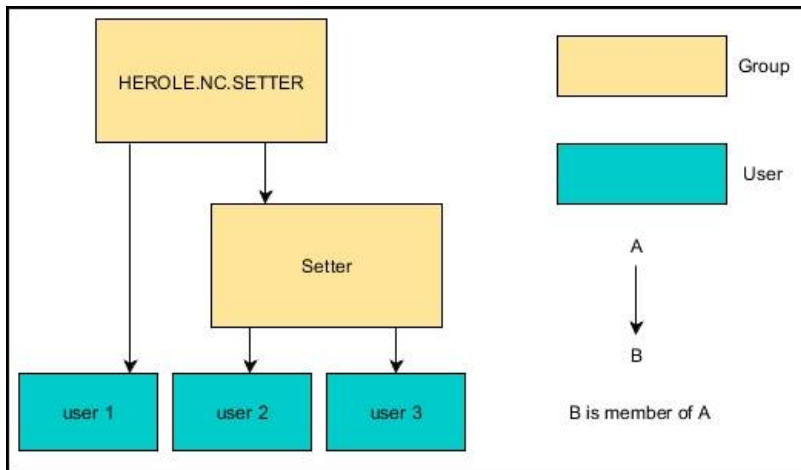
Example 2

Users from various sectors (workshops) are members of groups with different prefixes:
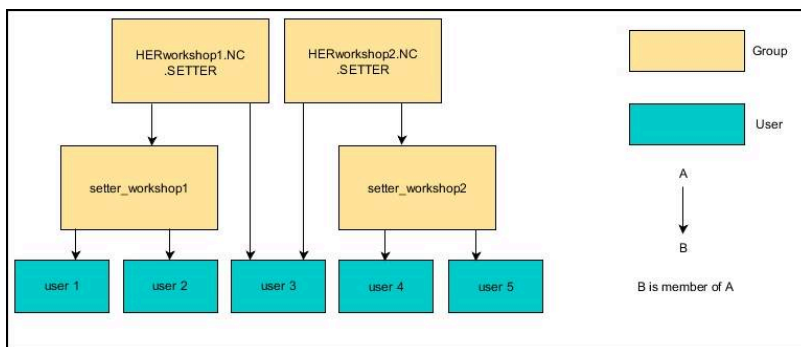
Exporting and importing a Windows configuration file

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.

Creating further users

Before you can create further users, the following requirements must be met:

◾ User administration has been configured
◾ LDAP database has been selected and configured

> ⓘ The **User administration** tab is only active for the following databases:
> ◾ **Local LDAP database**
> ◾ **LDAP on remote computer**
>
> If **Connection to Windows domain** is used, you need to configure the users in the Windows domain.
>
> **Further information:** "Logon to Windows domain", Page 740

Open the User administration tab

To manage users:

▶ Call user administration

▶ Select the **User administration** tab

▶ Press the **EDIT ON** soft key

> The control prompts you to enter the password of the user database, if applicable.

> After you have entered the password, the control opens the **User administration** menu.

You can edit the existing users or create new users.

Creating a new user

To create a new user:

▶ Press the **Create new user** soft key
> The control opens a window for creating a user.
▶ Enter the user name
▶ Enter the password for the user

ⓘ The user must change the password when logging in for the first time.
Further information: "Login to user administration", Page 760

▶ Create a user description, if desired
▶ Press the **Add role** soft key
▶ In the selection window, select the roles corresponding to the user
Further information: "Definition of roles", Page 751
▶ Press the **Add** soft key

ⓘ Two additional soft keys are available in the menu:
■ **Add external login**
Adds Remote.HEROS.Admin in place of HEROS.Admin, for example.
This role is only enabled for remote logon to the system.
■ **Add local login**
Adds Local.HEROS.Admin in place of HEROS.Admin, for example.
This role is only enabled for local logon at the control's screen.

▶ Press the **CLOSE** soft key
> The control closes the window for creating a user.
▶ Press the **OK** soft key
▶ Press the **APPLY** soft key
> The control adopts the changes.
▶ Press the **END** soft key
> The control closes user administration.

ⓘ If you have not restarted your control after configuring the database, the control prompts you to restart for the changes to take effect.
Further information: "Configuring the user administration", Page 734

Adding profile images

You can optionally also assign images to the users. The **Default user icons:** from HEIDENHAIN are available for this purpose. You can also load your own images in JPEG or PNG format to the control. Then you can use these image files as profile images.

To define profile images:

▶ Log on a user with the HEROS.Admin role, e.g. **useradmin**

   Further information: "Login to user administration", Page 760

▶ Call user administration

▶ Select the **User administration** tab

▶ Press the **Edit user** soft key

▶ Press the **Change icon** soft key

▶ Select the desired image in the menu

▶ Press the **Choose icon** soft key

▶ Press the **OK** soft key

▶ Press the **APPLY** soft key

> The control adopts the changes.

$\mathbf{i}$ You can also directly add profile images when creating the users.

Password settings in user administration

Users with the HEROS.Admin role can specify exact requirements for user passwords on the **Password settings** tab.
Further information: "Rights", Page 754

$\mathbf{i}$ If you do not comply with the requirements defined here when creating your password, the control will issue an error message.

To open the **Password settings** tab:

▶ Log on as a user with HEROS.Admin role

▶ Call user administration

▶ Select the **Password settings** tab

▶ Press the **EDIT ON** soft key

> The control opens the **Enter password of LDAP database** window.

▶ Enter password

> The control enables the **Password settings** tab for editing.

The control allows you to configure user password requirements by setting various parameters.

To edit the parameters:

▶ Open the **Password settings** tab

▶ Select the desired parameter

> The control highlights the selected parameter in blue.

▶ Set the desired parameter using the slider

> The control shows the selected parameter in the display field.
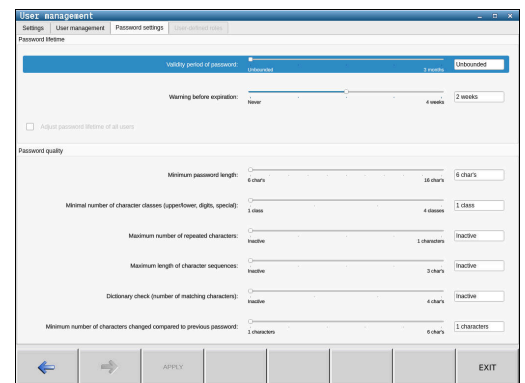
APPLY	▶ Press the **APPLY** soft key
	> The control applies the change.

The following parameters are available:

Password lifetime

■ **Validity period of password:**

Here, you can indicate how long the password can be used.

■ **Warning before expiration:**

From the defined time, a warning will be issued that the password will soon expire.

Password quality

■ **Minimum password length:**

Here, you can indicate the minimum password length.

■ **Minimal number of character classes (upper/lower, digits, special):**

Here, you can indicate the minimum number of different character classes required in the password.

■ **Maximum number of repeated characters:**

Here, you can indicate the maximum number of identical successive characters in the password.

■ **Maximum length of character sequences:**

Here, you can indicate the maximum length of the character sequences to be used in the password, e.g. 123.

■ **Dictionary check (number of matching characters):**

Here, you can enable a check whether the password contains known words and specify the allowed number of meaningful characters.

■ **Minimum number of characters changed compared to previous password:**

Here, you can specify how many characters in the new password must be different from the previous one.

Access rights

The user administration is based on the Unix rights management. Access to the control is controlled by means of rights.
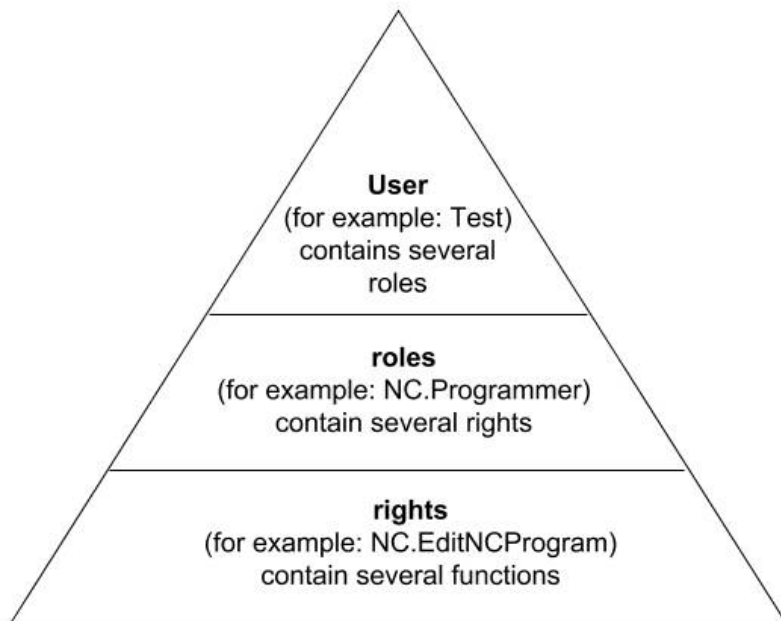
The user administration differentiates between the following terms:

- User
- Roles
- Rights

Roles

Roles consist of a combination of rights that cover certain functions provided by the control.

- **Operating system roles:**
- **NC operator roles:**
- **Machine tool builder (PLC) roles:**

Rights

Rights consist of a combination of functions that cover a task area of the control, e.g. editing the tool table.

- HEROS rights
- NC rights
- PLC rights (machine manufacturer)

If more than one role is assigned to a user, he will be granted all rights contained in these roles.

> Ensure that every user is assigned all access rights he needs. The access rights result from the actions a user performs on the control.

HEIDENHAIN function users

HEIDENHAIN function users are pre-defined users that are automatically created upon activation of user administration. Function users cannot be changed.

HEIDENHAIN provides four different function users in the control's factory default setting.

■ **oem**

The **oem** function user is intended for the machine manufacturer. The **oem** function user allows you to access the **PLC:** drive of the control.

■ **Function users defined by the machine manufacturer**

> ⚙ Refer to your machine manual.
>
> The users defined by the machine manufacturer can differ from those pre-defined by HEIDENHAIN.
>
> The machine manufacturer's function users can already be active in **legacy mode** and replace code numbers.
>
> By entering code numbers or passwords that replace code numbers, you can temporarily enable rights of **oem** function users.
>
> **Further information:** "Current User", Page 766

■ **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**.

■ **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.

Definition of roles

HEIDENHAIN combines several rights for individual task areas to roles. Different pre-defined roles that you can use to assign rights to your users are available. The tables below describe the individual rights of the different roles.

> **i** Every user should have at least one role from the operating system area and at least one role from the programming area.
>
> You can enable a role either for local login or for remote login. With local login, the user directly logs on to the control at the control's screen. A remote login (DNC) is a connection via SSH.
>
> You can therefore also make the rights of a user dependent on the access used to operate the control.
>
> If a role is only enabled for local login, Local. is added to the role name (e.g., Local.HEROS.Admin instead of HEROS.Admin).
>
> If a role is only enabled for remote login, Remote. is added to the role name (e.g., Remote.HEROS.Admin instead of HEROS.Admin).

Advantages of classification in roles:

- Simplified administration
- Different rights are compatible between different software versions of the control and different machine manufacturers.

> **i** Various applications require access to certain interfaces. Where required, the administrator also needs to set up rights to access the required interfaces as well as rights to access certain functions and additional programs. These rights are granted by the **Operating system roles:**.

> **i** The following contents can change in the following software versions of the control:
> - HEROS role names
> - Unix groups
> - Basic ID number

Operating system roles:

Role	Privileges		
	HEROS role name	**UNIX group**	**Basic ID number**
HEROS.RestrictedUser	Role for a user with minimum rights on the operating system.		
	■ HEROS.MountShares	■ mnt	■ 335
	■ HEROS.Printer	■ lp	■ 9
HEROS.NormalUser	Role for a normal user with limited rights on the operating system.		
	This role grants the rights of the RestrictedUser role, as well as the following rights:		
	■ HEROS.SetShares	■ mntcfg	■ 334
	■ HEROS.ControlFunctions	■ ctrlfct	■ 340
HEROS.LegacyUser	With the **LegacyUser** role, the behavior regarding the operating system of the control is identical to that of older software versions without user administration. User administration remains 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.VMSharedFolders	■ vboxsf	■ 1000
HEROS.LegacyUserNoCtrlfct	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
HEROS.Admin	The configuration of the network and the configuration of the user administration are some of the rights granted by this role.		
	This role grants the rights of the **LegacyUser** role, as well as the following rights:		
	■ HEROS.BackupMachine	■ backup	■ 338
	■ HEROS.UserAdmin	■ useradmin	■ 339

NC operator roles:

Role	Privileges		
	HEROS role name	**UNIX group**	**Basic ID number**
NC.Operator	This role allows you to run NC programs.		
	▪ NC.OPModeProgramRun	▪ NCOpPgmRun	▪ 302
NC.Programmer	This role grants the rights of NC programming.		
	This role grants the rights of the Operator role, as well as the following rights:		
	▪ NC.EditNCProgram	▪ NCEdNCProg	▪ 305
	▪ NC.EditPalletTable	▪ NCEdPal	▪ 309
	▪ NC.EditPresetTable	▪ NCEdPreset	▪ 308
	▪ NC.EditToolTable	▪ NCEdTool	▪ 306
	▪ NC.OPModeMDi	▪ NCOpMDI	▪ 301
	▪ NC.OPModeManual	▪ NCOpManual	▪ 300
NC.Setter	This role allows you to edit the pocket table.		
	This role grants the rights of the Programmer role, as well as the following rights:		
	▪ NC.ApproveFsAxis	▪ NCApproveFsAxis	▪ 319
	▪ NC.EditPocketTable	▪ NCEdPocket	▪ 307
	▪ NC.SetupDrive	▪ NCSetupDrv	▪ 315
	▪ NC.SetupProgramRun	▪ NCSetupPgRun	▪ 303
NC.AutoProductionSetter	This role allows you to execute all NC functions, including programming a scheduled NC program start.		
	This role grants the rights of the Setter role, as well as the following rights:		
	▪ NC.ScheduleProgramRun	▪ NCSchedulePgRun	▪ 304
NC.LegacyUser	With the **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.		
NC.AdvancedEdit	This role allows you to use special functions of the NC and table editors.		
	▪ Special functions of Q parameter programming and editing the table header		
	Replacement for code number **555343**		
	▪ NC.EditNCProgramAdv	▪ NCEditNCPgmAdv	▪ 327
	▪ NC.EditTableAdv	▪ NCEditTableAdv	▪ 328
NC.RemoteOperator	This role allows you to start NC programs from an external application.		
	▪ NC.RemoteProgramRun	▪ NCRemotePgmRun	▪ 329

Machine tool builder (PLC) roles:

Role	Privileges		
	HEROS role name	**UNIX group**	**Basic ID number**
PLC.ConfigureUser	This roles grants the rights on code number **123**.		
	▪ NC.ConfigUserAdv	▪ NCConfigUserAdv	▪ 316
	▪ NC.SetupDrive	▪ NCSetupDrv	▪ 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 tool builder (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

Rights

The table below lists all of the individual rights.

Rights:

HEROS role name	Description
HEROS.Printer	Data output to network printers
HEROS.PrinterAdmin	Configuration of network printers
NC.OPModeManual	Functions in **Machine** operating mode:
	▪ Running the machine manufacturer's macros
	▪ **TSF** menu
	▪ **Teach-in**
NC.OPModeMDi	Functions in **Machine** operating mode:
	▪ MDI cycles
	▪ DIN cycles
	▪ M functions
NC.OPModeProgramRun	Functions in **Machine** operating mode: **Program run**
NC.SetupProgramRun	Functions in **Machine** operating mode: Probing
NC.ScheduleProgramRun	Right not assigned
NC.EditNCProgram	Editing NC programs
NC.EditToolTable	Editing the tool data
	▪ Editing the tool table
	▪ Editing the tool holder table
	▪ Editing the technology table
NC.EditPocketTable	Edit Pocket Table

HEROS role name	Description
NC.EditPresetTable	Functions in **Machine** operating mode: ◾ Setting axis values ◾ Deleting offset
NC.EditPalletTable	Right not assigned
NC.SetupDrive	Offset adjustment for the analog axes
NC.ApproveFsAxis	Confirming test position of safe axes
NC.EditNCProgramAdv	Right not assigned
NC.EditTableAdv	Right not assigned
HEROS.SetTimezone	Adjustment of time and date, time zone, and time synchronization via NTP and HEROS menu
HEROS.SetShares	Configuration of network shares mounted by the control
HEROS.MountShares	Connecting and disconnecting network shares with the control
HEROS.SetNetwork	Configuration of network and relevant settings for data security
HEROS.BackupUsers	Data backup on the control—for all users configured on the control
HEROS.BackupMachine	Backup and restoring data of the entire machine configuration
HEROS.UserAdmin	Configuration of user administration on the control This includes creating, deleting, and configuring local users
HEROS.ControlFunctions	Control function of the operating system ◾ Auxiliary functions, such as starting and stopping NC software. ◾ Telemaintenance ◾ Advanced diagnostic functions, such as log data
HEROS.SWUpdate	Installation of software updates for the control
HEROS.VMSharedFolders	Access to shared folders of a virtual machine Only relevant when running a programming station within a virtual machine
NC.RemoteProgramRun	NC program start from an external application, e.g. via the DNC interface
NC.ConfigUserAdv	Configuration access to the contents that have been enabled through code number **123**
NC.Data.AccessServiceRead	Read-only access to the service data during servicing

Enabling Autologin

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.

For further authorizations, the control still requires an authentication to be entered.

In order to enable **Autologin**, the following requirements must be met:

- User administration has been configured
- The user for **Autologin** has been defined

To enable the **Autologin** function:

- ▶ Call user administration
- ▶ Select the **Settings** tab
- ▶ Press the **Global settings** soft key
- ▶ Select the **Enable autologin** check box
- > The control opens a window for selecting the user.
- ▶ Select the user
- ▶ Enter the user's password
- ▶ Press the **OK** soft key

User authentication from external applications

Introduction

If user administration is active, external applications also need to authenticate a user so that the appropriate rights can be assigned.

For DNC connections via 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.

$\mathbf{i}$ The encryption used with the SSH tunnel secure the communication from attackers.

**Principle of transmission through an SSH tunnel**

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

An SSH connection is always set up between an SSH client and an SSH server.

A key pair is used to secure 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 specific user.

The client tries to connect to the server by 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.
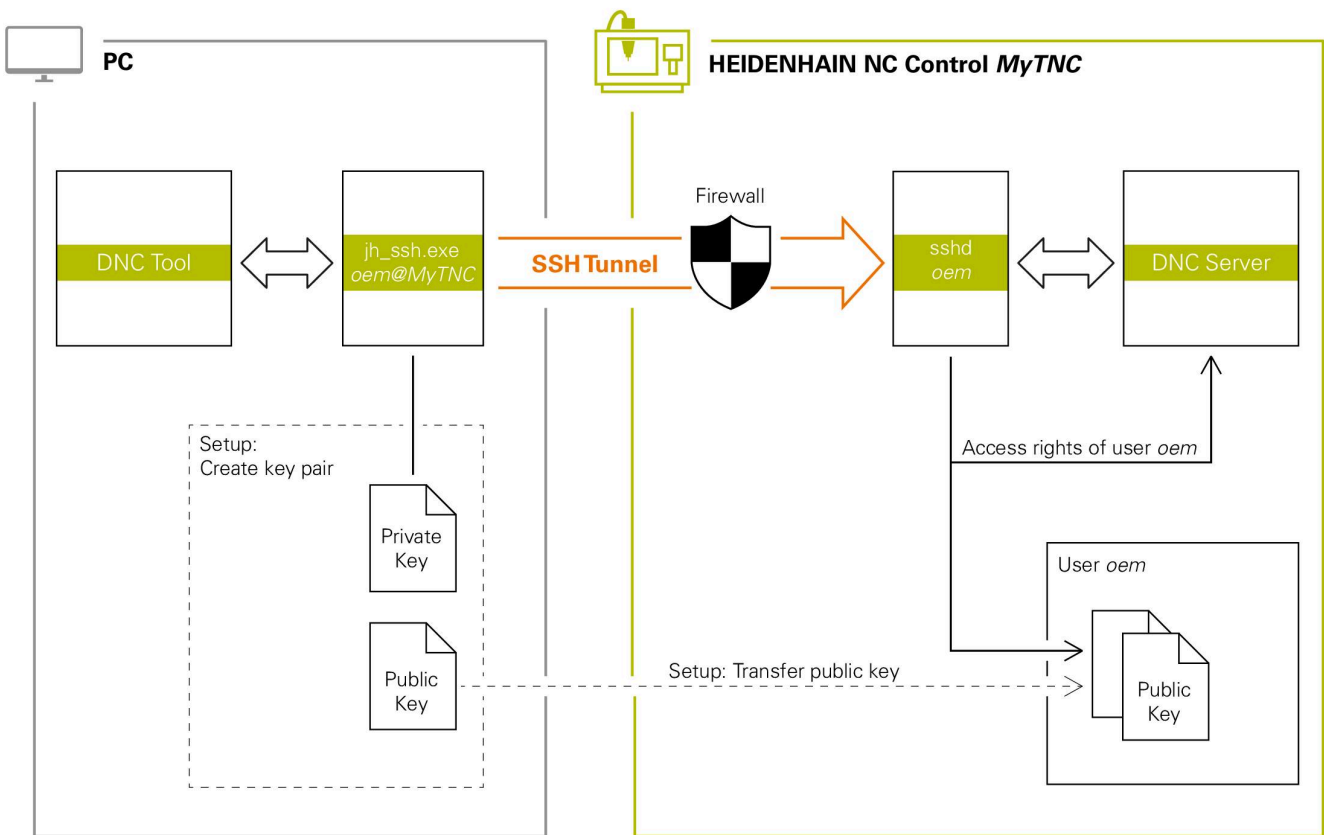
Use in external applications

ℹ️ When user administration is active, you can set up only secure network connections via SSH. The control automatically disables the LSV2 connections via the serial interfaces (COM1 and COM2) and the network connections without user authentication.

If user administration is inactive, the control also automatically blocks insecure 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 insecure connections. These machine parameters are contained in the **CfgDncAllowUnsecur** (135400) data object.

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.

ℹ️ Once the connection configurations have been set up, they can be shared among all HEIDENHAIN PC tools for building up a connection.

This also applies to applications that are using the HEIDENHAIN DNC component from RemoTools SDK for communication. There is no need to adapt existing customer applications.

ℹ️ In order to expand the connection configuration by 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.

Setting up and removing a secure connection

To set up a secure connection for the logged-on user:

▶ Press the **DIADUR** key to open the **HEROS menu**
▶ Select the **Settings** menu item
▶ Select the **Current User** menu icon
▶ Press the **Certificate and keys** soft key
> The control opens the **Certificate and keys** window.
▶ Select the **Allow password authentication** function
▶ Press the **Store and restart server now** soft key
▶ Use **TNCremo** to set up the secure connection (TCP secure).

> 📖 For details, please refer to the integrated help system of TNCremo.

> TNCremo has stored the public key on the control.

> ℹ In order to ensure maximum security, deactivate the **Allow password authentication** function after the public key has been stored.

▶ Deselect the **Allow password authentication** function
▶ Press the **Store and restart server now** soft key
> The control has adopted your changes.

> ℹ ▪ In addition to using the PC tools for setup with password authentication, you can also import the public key into the control by using a USB flash drive or a 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.

To delete a key in the control so that a user can no longer use the secure connection:

▶ Press the **DIADUR** key to open the **HEROS menu**
▶ Select the **Settings** menu item
▶ Select the **Current User** menu icon
▶ Press the **Certificate and keys** soft key
> The control opens the **Certificate and keys** window.
▶ Select the key to be deleted
▶ Press the **Delete SSH key** soft key
> The control deletes the selected key.

Disabling non-secure connections in the firewall

The DNC protocols LSV2 and RPC should be blocked in the firewall to ensure that the use of secure connections provides a real IT security advantage for the control.

This requires that the following parties change to secure connections:

■ Machine manufacturers with all external applications (e.g., pick-and-place robots)

> ℹ️   If the additional application is connected via **machine network X116**, there is no need to switch to an encrypted connection.

■ Users with custom external applications

If you have ensured that the connections of all parties are secure, the LSV2 and RPC DNC protocols can be blocked in the **Firewall**.

To block the protocols in the firewall:

▶ Press the **DIADUR** key to open the **HEROS menu**
▶ Select the **Settings** menu item
▶ Select the **Firewall** menu item
▶ Select the **Prohibit all** method for **DNC** and **LSV2**
▶ Select the **Apply** function
> The control saves your changes.
▶ Close the window with **OK**

Login to user administration

The control displays the Login dialog in the following cases:

■ After the **User logout** function has been executed
■ After the **Switch user** function has been executed
■ After the screen has been locked by the **screensaver**
■ Immediately after control startup if user administration is active and **Autologin** is not enabled

The login dialog provides the following options:

■ Users who logged in at least once
■ **Other** user
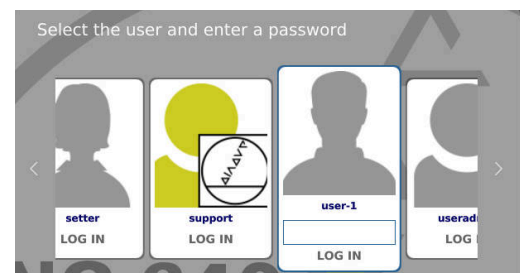
Logging in a user for the first time

To log in as a user for the first time, you need to use the **Other** input field.

To log in as a user with the **Other** option for 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

> The control opens a window with the **Password expired** message. **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 user is shown in the login dialog.

Logging in a known user with password

To log in as a user who is displayed in the login dialog, proceed as follows:

▶ Select the respective user in the login dialog

> The control enlarges the user icon you selected.

▶ Enter the user password

> The control uses the selected user to log you in.

❶ The control shows in the Login dialog whether CAPS LOCK is active.

Logging in a user with token

To log in as a user with a token, proceed as follows:

▶ Hold the token close to the reader

▶ Enter PIN, if required

> The control uses the selected user to log you in.

▶ Remove the token from the reader

Password requirements

> ℹ️ For reasons of security, passwords should comply with the following criteria:
> - Eight characters minimum
> - Letters, numbers, and special characters
> - Avoid using whole words or a sequence of characters (e.g., Anna or 123)

Keep in mind that an administrator can define the requirements to be met by a password. The password requirements include:
- Minimum length
- Minimum number of different types of characters
  - Capital letters
  - Small letters
  - Numbers
  - Special characters
- Maximum length of character strings (e.g., 54321 = string of five characters)
- The number of characters must match the dictionary check
- Minimum number of characters changed compared to previous password

If the new password does not meet the requirements, an error message will be issued. This means that you have to enter a different password.

> ℹ️ Administrators can specify the expiration of passwords. If you do not change your password within the valid period, you will no longer be able to log on as that user. In this case, an administrator needs to reset the user password before you can log on again.
> ▶ Change your password at regular intervals
>   "Changing the current user's password"
> ▶ Take note of the password change warnings

Switching or logging out users

Use the **Switch-off** HEROS menu item or the icon of the same name at bottom right in the menu bar to open the **Switch off and restart** window.

The control provides the following options:

- **Switch-off**:
    - The control stops and ends all additional programs and functions
    - The system is shut down
    - The control is switched off
- **Restart**:
    - The control stops and ends all additional programs and functions
    - The system is restarted
- **Log out**:
    - The control ends all additional programs
    - The user is logged out
    - The login window appears

> **i** To continue, you need to log in a new user and enter his password.
>
> The NC program continues running under the previously logged-on user.

- **Switch user**:
    - The login window appears
    - The user is not logged out

> **i** You can close the login window with **Cancel** without entering a password.
>
> All additional programs and the NC programs started by the logged-on user continue running.

Screensaver with lockout

You can use the screensaver to lock the control. NC programs that have already been started continue running during this period.

> **i** You need to enter a password to unlock the screensaver again.
>
> **Further information:** "Login to user administration", Page 760
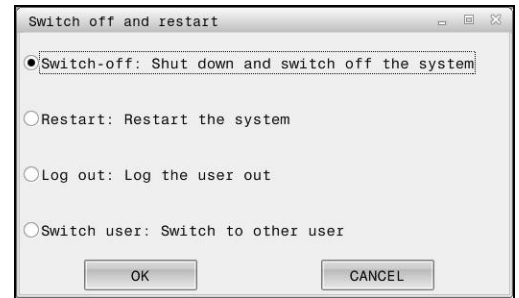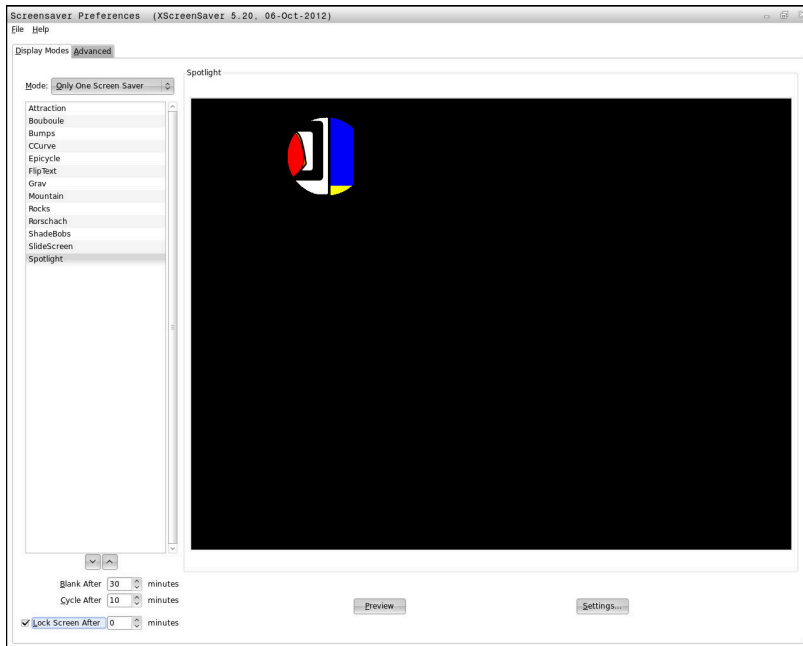
To access the screensaver settings in the **HEROS menu**:

▶ Press the **DIADUR** key to open the **HEROS menu**

▶ Select the **Settings** menu item

▶ Select the **Screensaver** menu item

The screensaver provides the following options:

◼ Use the **Blank After** setting to define after how many minutes the screensaver should turn on.

◼ Use the **Lock Screen After** setting to activate the lock with password protection.

◼ The time value behind **Lock screen after** defines the delay after which the lock will take effect after the screensaver has been turned on. A value of **0** means that the lock becomes effective immediately after the screensaver has been turned on.

When the lock is active and you use one of the input devices (for example, by moving the mouse), the screensaver disappears. The control displays a lock screen instead.
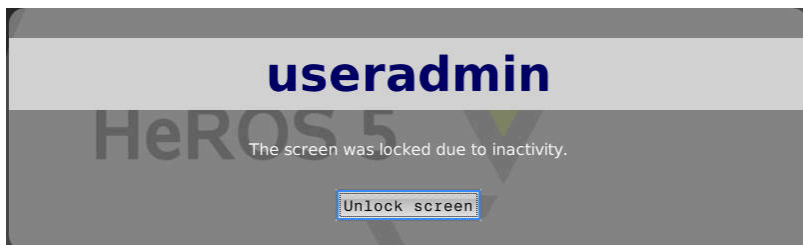
Use **Unlock screen** or **Enter** to open the login window again.

HOME directory

When user administration is active, a private **HOME:** directory, to which you can save your private programs and files, is available to every user.

The **HOME:** directory can be viewed by the respectively logged-in users as well as users with the HEROS.Admin role.
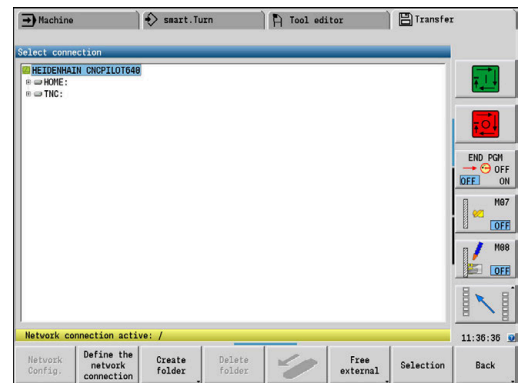
Current User

With **Current User**, you can view the group rights of the currently logged-on user in the **HEROS** menu.

> ℹ️ 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:** "HEIDENHAIN function users", Page 750

To call the **Current User**:

▶ Press the **DIADUR** key to open the **HEROS menu**

▶ Select the **Settings** menu item

▶ Select the **Current User** menu icon

User administration allows you to temporarily increase the current user's rights by adding the rights of a user you selected.

To temporarily increase a user's rights:

▶ Call **Current User**

▶ Press the **Add rights** soft key

▶ Select the user

▶ Enter the user's user name

▶ Enter the selected user's password

> The control temporarily increases the logged-on user's rights by adding the rights of the user specified in **Add rights**.

> ℹ️ You can temporarily enable the rights of **oem** function users. For this purpose, enter the appropriate code number or the password defined by the machine manufacturer.

You have the following options to remove the rights that have temporarily been added:

- Enter the code number **0**
- User logout
- Press the **Remove added rights** soft key

To select the **Remove added rights** soft key:

▶ Call **Current User**

▶ Select the **Added rights** tab

▶ Press the **Remove added rights** soft key
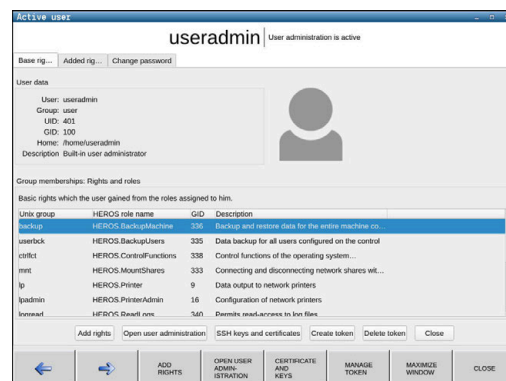
Changing the current user's password

In the **Current User** menu item, you can change the current user's password.

To change the current user's password:

▶ Call **Current User**

▶ Select the **Change password** tab

▶ Enter the old password

▶ Press the **Verify old password** soft key

> The control checks whether you have correctly entered your old password.

> If the password is correct, the fields **New password** and **Repeat password** are enabled.

▶ Enter a new password

▶ Repeat the new password

▶ Press the **Set new password** soft key

> The control compares the requirements defined for passwords by the administrator to the password you entered.

  Further information: "Login to user administration", Page 760

> The message **Password was changed successfully** appears.

Defining login with token

The control also permits login with a token. This ensures secure login without the user having to enter a password.

⚙ Refer to your machine manual.

  The machine manufacturer must prepare the machine for use with a token. For example, an appropriate reader must be mounted on the machine.

In the **Current User** menu item, you can define login with a token for the current user.

To create a token:

▶ Call **Current User**

▶ Select **Create token**

▶ Select the type of token with **Switch type**, if required

▶ Enter the user's password

▶ Enter PIN, if required

▶ Hold the token close to the reader

▶ Select **Reload list**

▶ Choose a token from the list

▶ Select **Start writing**

▶ Enter PIN, if defined

> The control starts the write process.

▶ Hold the token close to the reader until the end of the write process

> When writing is complete, the control shows a message.

Delete token allows you to delete the created token and use password entry again.

Dialog for requesting additional rights

If you do not have the rights required for a specific **HEROS menu** item, the control opens the window for requesting additional rights:

In this window, you can temporarily obtain more rights by adding another user's rights.

In the **Users that have this right:** field, the control lists all existing users that have the right to use this function.

> ℹ️ If **Connection to Windows domain** is used, only users that were recently logged on are shown in the selection menu.
>
> To attain the rights of users that are not shown, enter their user data. The control will then recognize those users that are contained in the user database.

Obtaining more rights

To temporarily add another user's rights to your rights:

▶ Select the user whose rights you need

▶ Enter the user's name

▶ Enter the user's password

▶ Press the **Set right** soft key

> The control adds the entered user's rights to your rights.

Further information: "Current User", Page 766
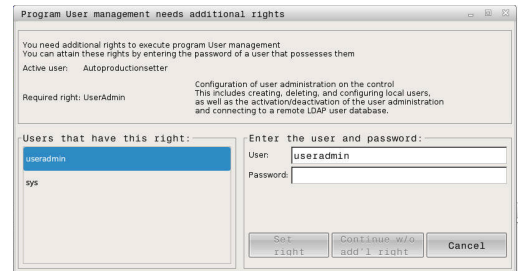
12.11 Changing the HEROS conversational language

The HEROS conversational language is internally based on the NC conversational language. Therefore, the **HEROS menu** and the control cannot permanently be set to two different conversational languages.

When you change the NC conversational language, the HEROS conversational language will only reflect this change after a restart of the control.

ℹ️ With the optional machine parameter **applyCfgLanguage** (no. 101305), you define the behavior of the control if the NC conversational language and in the HEROS conversational language do not match.

The following link provides information on how to change the NC conversational language:

Further information: "Parameter editor", Page 618

Changing the language layout of the keyboard

You can change the language layout of the keyboard for HEROS applications.

To change the language layout of the keyboard for HEROS applications:

▶ Select the HEROS menu icon
▶ Select **Settings**
▶ Select **Language/Keyboards**
> The control opens the **helocale** window.
▶ Select the **Keyboards** tab
▶ Select the desired keyboard layout
▶ Select **Apply**
▶ Press **OK**
▶ Select **Apply**
> The changes are applied.

13

**Tables and
Overviews**

13.1 Thread

Thread parameters

To determine the thread parameters, the control uses the following table.

Where:

- **F**: **Thread pitch** is calculated from the diameter, depending on the thread type, where an asterisk ***** is given
  Further information: "Thread", Page 772
- **P**: **Thread depth**
- **R**: **Thread width**
- **A**: **Thread angle, left**
- **W**: **Thread angle, right**

Calculation formula: **Kb** = 0.26384 * **F** − 0.1 * √ **F**

Thread backlash **ac** (depending on the **Thread pitch**):

- **Thread pitch** <= 1 : **ac** = 0.15
- **Thread pitch** <= 2 : **ac** = 0.25
- **Thread pitch** <= 6 : **ac** = 0.5
- **Thread pitch** <= 13 : **ac** = 1

Sort of thread Q		F	P	R	A	W
Q = 1 Metric ISO fine-pitch thread	External	−	0.61343 * **F**	**F**	30°	30°
	Internal	−	0.54127 * **F**	**F**	30°	30°
Q = 2 Metric ISO thread	External	*	0.61343 * **F**	**F**	30°	30°
	Internal	*	0.54127 * **F**	**F**	30°	30°
Q = 3 Metric ISO tapered thread	External	−	0.61343 * **F**	**F**	30°	30°
Q = 4 Metric ISO tapered fine-pitch thread		−	0.61343 * **F**	**F**	30°	30°
Q = 5 Metric ISO trapezoid thread	External	−	0.5 * **F** + ac	0.633 * **F**	15°	15°
	Internal	−	0.5 * **F** + **ac**	0.633 * **F**	15°	15°
Q = 6 Flat metric trapezoid thread	External	−	0.3 * **F** + **ac**	0.527 * **F**	15°	15°
	Internal	−	0.3 * **F** + **ac**	0.527 * **F**	15°	15°
Q = 7 Metric buttress thread	External	−	0.86777 * **F**	0.73616 * **F**	3°	30°
	Internal	−	0.75 * **F**	**F** − **Kb**	30°	3°
Q = 8 Cylindrical round thread	External	*	0.5 * **F**	**F**	15°	15°
	Internal	*	0.5 * **F**	**F**	15°	15°
Q = 9 Cylindrical Whitworth thread	External	*	0.64033 * **F**	**F**	27.5°	27.5°
	Internal	*	0.64033 * **F**	**F**	27.5°	27.5°
Q = 10 Tapered Whitworth thread	External	*	0.640327 * **F**	**F**	27.5°	27.5°
Q = 11 Whitworth pipe thread	External	*	0.640327 * **F**	**F**	27.5°	27.5°
	Internal	*	0.640327 * **F**	**F**	27.5°	27.5°
Q = 12 Nonstandard thread		−	−	−	−	−
Q = 13 UNC US coarse thread	External	*	0.61343 * **F**	**F**	30°	30°
	Internal	*	0.54127 * **F**	**F**	30°	30°

Sort of thread Q		F	P	R	A	W
Q = 14 UNF US fine-pitch thread	External	*	0.61343 * **F**	**F**	30°	30°
	Internal	*	0.54127 * **F**	**F**	30°	30°
Q = 15 UNEF US extra-fine-pitch thread	External	*	0.61343 * **F**	**F**	30°	30°
	Internal	*	0.54127 * **F**	**F**	30°	30°
Q = 16 NPT US taper pipe thread	External	*	0.8 * **F**	**F**	30°	30°
	Internal	*	0.8 * **F**	**F**	30°	30°
Q = 17 NPTF US taper dryseal pipe thread	External	*	0.8 * **F**	**F**	30°	30°
	Internal	*	0.8 * **F**	**F**	30°	30°
Q = 18 NPSC U.S. cylindrical pipe thread with lubricant	External	*	0.8 * **F**	**F**	30°	30°
	Internal	*	0.8 * **F**	**F**	30°	30°
Q = 19 NPFS U.S. cylindrical pipe thread without lubricant	External	*	0.8 * **F**	**F**	30°	30°
	Internal	*	0.8 * **F**	**F**	30°	30°

Thread pitch

Q = 2
Metric ISO thread

Diameter (in mm)	Thread pitch
1	0.25
1.1	0.25
1.2	0.25
1.4	0.3
1.6	0.35
1.8	0.35
2	0.4
2.2	0.45
2.5	0.45
3	0.5
3.5	0.6
4	0.7
4.5	0.75
5	0.8
6	1
7	1
8	1.25
9	1.25
10	1.5
11	1.5
12	1.75
14	2
16	2
18	2.5
20	2.5
22	2.5
24	3
27	3
30	3.5
33	3.5
36	4
39	4
42	4.5
45	4.5
48	5
52	5

Diameter (in mm)	Thread pitch
56	5.5
60	5.5
64	6
68	6

Q = 8
Cylindrical round thread

Diameter (in mm)	Thread pitch
12	2.54
14	3.175
40	4.233
105	6.35
200	6.35

Q = 9
Cylindrical Whitworth thread

Thread designation	Diameter (in mm)	Thread pitch
1/4"	6.35	1.27
5/16"	7.938	1.411
3/8"	9.525	1.588
7/16"	11.113	1.814
1/2"	12.7	2.117
5/8"	15.876	2.309
3/4"	19.051	2.54
7/8"	22.226	2.822
1"	25.401	3.175
1 1/8"	28.576	3.629
1 1/4"	31.751	3.629
1 3/8"	34.926	4.233
1 1/2"	38.101	4.233
1 5/8"	41.277	5.08
1 3/4"	44.452	5.08
1 7/8"	47.627	5.645
2"	50.802	5.645
2 1/4"	57.152	6.35
2 1/2"	63.502	6.35
2 3/4"	69.853	7.257

Q = 10
Tapered Whitworth thread

Thread designation	Diameter (in mm)	Thread pitch
1/16"	7.723	0.907
1/8"	9.728	0.907
1/4"	13.157	1.337
3/8"	16.662	1.337
1/2"	20.995	1.814
3/4"	26.441	1.814
1"	33.249	2.309
1 1/4"	41.91	2.309
1 1/2"	47.803	2.309
2"	59.614	2.309
2 1/2"	75.184	2.309
3"	87.884	2.309
4"	113.03	2.309
5"	138.43	2.309
6"	163.83	2.309

Q = 11
Whitworth pipe thread

Thread designation	Diameter (in mm)	Thread pitch
1/8"	9.728	0.907
1/4"	13.157	1.337
3/8"	16.662	1.337
1/2"	20.995	1.814
5/8"	22.911	1.814
3/4"	26.441	1.814
7/8"	30.201	1.814
1"	33.249	2.309
1 1/8"	37.897	2.309
1 1/4"	41.91	2.309
1 3/8"	44.323	2.309
1 1/2"	47.803	2.309
1 3/4"	53.746	1.814
2"	59.614	2.309
2 1/4"	65.71	2.309
2 1/2"	75.184	2.309
2 3/4"	81.534	2.309
3"	87.884	2.309
3 1/4"	93.98	2.309
3 1/2"	100.33	2.309
3 3/4"	106.68	2.309
4"	113.03	2.309
4 1/2"	125.73	2.309
5"	138.43	2.309
5 1/2"	151.13	2.309
6"	163.83	2.309

Q = 13
UNC US coarse thread

Thread designation	Diameter (in mm)	Thread pitch
0.073"	1.8542	0.396875
0.086"	2.1844	0.453571428
0.099"	2.5146	0.529166666
0.112"	2.8448	0.635
0.125"	3.175	0.635
0.138"	3.5052	0.79375
0.164"	4.1656	0.79375
0.19"	4.826	1.058333333
0.216"	5.4864	1.058333333
1/4"	6.35	1.27
5/16"	7.9375	1.411111111
3/8"	9.525	1.5875
7/16"	11.1125	1.814285714
1/2"	12.7	1.953846154
9/16"	14.2875	2.116666667
5/8"	15.875	2.309090909
3/4"	19.05	2.54
7/8"	22.225	2.822222222
1"	25.4	3.175
1 1/8"	28.575	3.628571429
1 1/4"	31.75	3.628571429
1 3/8"	34.925	4.233333333
1 1/2"	38.1	4.233333333
1 3/4"	44.45	5.08
2"	50.8	5.644444444
2 1/4"	57.15	5.644444444
2 1/2"	63.5	6.35
2 3/4"	69.85	6.35
3"	76.2	6.35
3 1/4"	82.55	6.35
3 1/2"	88.9	6.35
3 3/4"	95.25	6.35
4"	101.6	6.35

Q = 14
UNF US fine-pitch thread

Thread designation	Diameter (in mm)	Thread pitch
0.06"	1.524	0.3175
0.073"	1.8542	0.352777777
0.086"	2.1844	0.396875
0.099"	2.5146	0.453571428
0.112"	2.8448	0.529166666
0.125"	3.175	0.577272727
0.138"	3.5052	0.635
0.164"	4.1656	0.705555555
0.19"	4.826	0.79375
0.216"	5.4864	0.907142857
1/4"	6.35	0.907142857
5/16"	7.9375	1.058333333
3/8"	9.525	1.058333333
7/16"	11.1125	1.27
1/2"	12.7	1.27
9/16"	14.2875	1.411111111
5/8"	15.875	1.411111111
3/4"	19.05	1.5875
7/8"	22.225	1.814285714
1"	25.4	1.814285714
1 1/8"	28.575	2.116666667
1 1/4"	31.75	2.116666667
1 3/8"	34.925	2.116666667
1 1/2"	38.1	2.116666667

Q = 15
UNEF US extra-fine-pitch thread

Thread designation	Diameter (in mm)	Thread pitch
0.216"	5.4864	0.79375
1/4"	6.35	0.79375
5/16"	7.9375	0.79375
3/8"	9.525	0.79375
7/16"	11.1125	0.907142857
1/2"	12.7	0.907142857
9/16"	14.2875	1.058333333
5/8"	15.875	1.058333333
11/16"	17.4625	1.058333333
3/4"	19.05	1.27
13/16"	20.6375	1.27
7/8"	22.225	1.27
15/16"	23.8125	1.27
1"	25.4	1.27
1 1/16"	26.9875	1.411111111
1 1/8"	28.575	1.411111111
1 3/16"	30.1625	1.411111111
1 1/4"	31.75	1.411111111
1 5/16"	33.3375	1.411111111
1 3/8"	34.925	1.411111111
1 7/16"	36.5125	1.411111111
1 1/2"	38.1	1.411111111
1 9/16"	39.6875	1.411111111
1 5/8"	41.275	1.411111111
1 11/16"	42.8625	1.411111111
1 3/4"	44.45	1.5875
2"	50.8	1.5875

Q = 16
NPT US taper pipe thread

Thread designation	Diameter (in mm)	Thread pitch
1/16"	7.938	0.94074074
1/8"	10.287	0.94074074
1/4"	13.716	1.411111111
3/8"	17.145	1.411111111
1/2"	21.336	1.814285714
3/4"	26.67	1.814285714
1"	33.401	2.208695652
1 1/4"	42.164	2.208695652
1 1/2"	48.26	2.208695652
2"	60.325	2.208695652
2 1/2"	73.025	3.175
3"	88.9	3.175
3 1/2"	101.6	3.175
4"	114.3	3.175
5"	141.3	3.175
6"	168.275	3.175
8"	219.075	3.175
10"	273.05	3.175
12"	323.85	3.175
14"	355.6	3.175
16"	406.4	3.175
18"	457.2	3.175
20"	508	3.175
24"	609.6	3.175

Q = 17
NPTF US taper dryseal pipe thread

Thread designation	Diameter (in mm)	Thread pitch
1/16"	7.938	0.94074074
1/8"	10.287	0.94074074
1/4"	13.716	1.411111111
3/8"	17.145	1.411111111
1/2"	21.336	1.814285714
3/4"	26.67	1.814285714
1"	33.401	2.208695652
1 1/4"	42.164	2.208695652
1 1/2"	48.26	2.208695652
2"	60.325	2.208695652
2 1/2"	73.025	3.175
3"	88.9	3.175

Q = 18
NPSC U.S. cylindrical pipe thread
with lubricant

Thread designation	Diameter (in mm)	Thread pitch
1/8"	10.287	0.94074074
1/4"	13.716	1.411111111
3/8"	17.145	1.411111111
1/2"	21.336	1.814285714
3/4"	26.67	1.814285714
1"	33.401	2.208695652
1 1/4"	42.164	2.208695652
1 1/2"	48.26	2.208695652
2"	60.325	2.208695652
2 1/2"	73.025	3.175
3"	88.9	3.175
3 1/2"	101.6	3.175
4"	114.3	3.175

Q = 19
NPFS U.S. cylindrical pipe thread
without lubricant

Thread designation	Diameter (in mm)	Thread pitch
1/16"	7.938	0.94074074
1/8"	10.287	0.94074074
1/4"	13.716	1.411111111
3/8"	17.145	1.411111111
1/2"	21.336	1.814285714
3/4"	26.67	1.814285714
1"	33.401	2.208695652

13.2 Undercut parameters

DIN 76 – undercut parameters

The control determines the parameters for the thread undercut (**Undercut DIN 76**) from the thread pitch. The undercut parameters are in accordance with DIN 13 for metric threads.

External thread

Thread pitch	I	K	R	W
0.2	0.3	0.7	0.1	30°
0.25	0.4	0.9	0.12	30°
0.3	0.5	1.05	0.16	30°
0.35	0.6	1.2	0.16	30°
0.4	0.7	1.4	0.2	30°
0.45	0.7	1.6	0.2	30°
0.5	0.8	1.75	0.2	30°
0.6	1	2.1	0.4	30°
0.7	1.1	2.45	0.4	30°
0.75	1.2	2.6	0.4	30°
0.8	1.3	2.8	0.4	30°
1	1.6	3.5	0.6	30°
1.25	2	4.4	0.6	30°
1.5	2.3	5.2	0.8	30°
1.75	2.6	6.1	1	30°
2	3	7	1	30°
2.5	3.6	8.7	1.2	30°
3	4.4	10.5	1.6	30°
3.5	5	12	1.6	30°
4	5.7	14	2	30°
4.5	6.4	16	2	30°
5	7	17.5	2.5	30°
5.5	7.7	19	3.2	30°
6	8.3	21	3.2	30°

Internal thread

Thread pitch	I	K	R	W
0.2	0.1	1.2	0.1	30°
0.25	0.1	1.4	0.12	30°
0.3	0.1	1.6	0.16	30°
0.35	0.2	1.9	0.16	30°
0.4	0.2	2.2	0.2	30°
0.45	0.2	2.4	0.2	30°
0.5	0.3	2.7	0.2	30°
0.6	0.3	3.3	0.4	30°
0.7	0.3	3.8	0.4	30°
0.75	0.3	4	0.4	30°
0.8	0.3	4.2	0.4	30°
1	0.5	5.2	0.6	30°
1.25	0.5	6.7	0.6	30°
1.5	0.5	7.8	0.8	30°
1.75	0.5	9.1	1	30°
2	0.5	10.3	1	30°
2.5	0.5	13	1.2	30°
3	0.5	15.2	1.6	30°
3.5	0.5	17.7	1.6	30°
4	0.5	20	2	30°
4.5	0.5	23	2	30°
5	0.5	26	2.5	30°
5.5	0.5	28	3.2	30°
6	0.5	30	3.2	30°

For internal threads, the control calculates the depth of the thread undercut according to the following formula:

Undercut depth = (N + I - K) / 2

Where:

- **I**: **Undercut depth**
- **K**: **Undercut width**
- **R**: **Undercut radius**
- **W**: **Undercut angle**
- **N**: **Nominal diameter of the thread**
- **I**: From the table
- **K**: **Core diameter of the thread**

DIN 509 E – undercut parameters

Diameter	I	K	R	W
<= 1.6	0.1	0.5	0.1	15°
> 1.6 – 3	0.1	1	0.2	15°
> 3 – 10	0.2	2	0.2	15°
> 10 – 18	0.2	2	0.6	15°
> 18 – 80	0.3	2.5	0.6	15°
> 80	0.4	4	1	15°

The undercut parameters are determined from the cylinder diameter.

Where:

- **I**: **Undercut depth**
- **K**: **Undercut width**
- **R**: **Undercut radius**
- **W**: **Undercut angle**

DIN 509 F – undercut parameters

Diameter	I	K	R	W	P	A
<= 1.6	0.1	0.5	0.1	15°	0.1	8°
> 1.6 – 3	0.1	1	0.2	15°	0.1	8°
> 3 – 10	0.2	2	0.2	15°	0.1	8°
> 10 – 18	0.2	2	0.6	15°	0.1	8°
> 18 – 80	0.3	2.5	0.6	15°	0.2	8°
> 80	0.4	4	1	15°	0.3	8°

The undercut parameters are determined from the cylinder diameter.

Where:

- **I**: **Undercut depth**
- **K**: **Undercut width**
- **R**: **Undercut radius**
- **W**: **Undercut angle**
- **P**: **Trav.dpth**
- **A**: **Trav.angle**

13.3 Technical information

Specifications

Components	■ MC 8420 T main computer with ■ CC 3xx controller unit or UEC 3xx ■ 15.6-inch multitouch screen
Operating system	■ Real-time HEROS 5 operating system (HEIDENHAIN Realtime Operating System) for machine control
Memory	■ 7.7 GB for NC programs
Input resolution and display step	■ X axis: 0.5 µm, diameter: 1 µm ■ Z and Y axes: 1 µm ■ U, V and W axes: 1 µm ■ C axis: 0.001° ■ B axis: 0.001°
Interpolation	■ Straight line: in 2 principal axes (max. ±100 m), in 3 principal axes (software option 70) ■ Circle: in 2 axes (max. radius: 999 m); additional linear interpolation of the third axis (software option 55 or software option 70) ■ C axis: interpolation in the linear axes X and Z with the C axis (software option 55) ■ B axis: 5-axis interpolation between X, Z, Y, B, and C axes (software option 54)
Feed rate	■ mm/min or mm/rev ■ Constant surface speed ■ Max. feed rate (60 000/pole pairs × ball screw pitch) at fPWM = 5000 Hz
Spindle	■ Max. 60 000 rpm (with one pole pair) ■ Max. 120 000 rpm (software option 49)
Axis feedback control	■ Integrated digital drive control for synchronous and asynchronous motors ■ Path interpolation: 3 ms ■ Fine interpolation: 0.2 ms, 0.1 ms (software option 49) ■ Position loop pulse: 0.2 ms, 0.1 ms (software option 49) ■ Speed loop pulse: 0.2 ms, 0.1 ms (software option 49)
Error compensation	■ Linear and nonlinear axis error ■ Backlash ■ Reversal spikes during circular movements ■ Hysteresis ■ Thermal expansion ■ Static and sliding friction
Data interfaces	■ Two Ethernet 1000BASE-T
Diagnostics	■ Fast and simple troubleshooting through integrated diagnostic aids
Ambient temperature	■ Operation: 5 °C to 40 °C ■ Storage: -20 °C to +60 °C

User functions

Configuration	■ Basic version: X and Z axis, spindle
	■ Driven tool
	■ C axis (option 55)
	■ Y axis (option 70)
	■ B axis (option 54)
	■ Parallel axes U, V, W (option 94)
	■ Digital current and speed control
	■ Rear-face machining with the counter spindle (option 132)
Machine operating mode	■ Manual slide movement through manual direction keys or electronic handwheel
	■ Graphic support for entering and running Teach-in cycles without saving the machining steps in alternation with manual machine operation
	■ Thread reworking (thread repair) in a second workpiece setup (option 11)
Teach-in submode (option 8)	■ Sequential linking of Teach-in cycles, where each machining cycle is run immediately after input, or is graphically simulated and subsequently saved.
Program run submode	All are possible in Single-Block and Full-Sequence modes:
	■ DIN PLUS programs
	■ smart.Turn programs (option 9)
	■ Teach-in programs (option 8)
Setup functions	■ Setting the workpiece datum
	■ Defining the tool change point
	■ Defining the protection zone
	■ Definition of machine dimensions
	■ Manual programs
	■ Tool measurement with probe or optics (option 17)

User functions

Programming – cycle programming (option 8)	■ Turning cycles for simple and complex contours, and contours described with ICP
	■ Contour-parallel turning cycles
	■ Recessing cycles for simple or complex contours, as well as contours defined with ICP
	■ Repetitions with recessing cycles
	■ Recess turning cycles for simple and complex contours, and contours described with ICP
	■ Undercut and parting cycles
	■ Engraving cycles
	■ Threading cycles for single or multi-start longitudinal, taper or API threads
	■ Cycles for axial and radial drilling, deep drilling, and tapping operations with the C axis (option 55)
	■ Thread milling with the C axis (option 55)
	■ Axial and radial milling cycles for slots, figures, single surfaces, and centric polygon surfaces, and for complex contours described with ICP for machining with the C axis (option 55)
	■ Helical slot milling with the C axis (option 55)
	■ Deburring of ICP contours (option 55)
	■ Linear and circular patterns for boring, drilling, and milling operations with the C axis (option 55)
	■ Context-sensitive help graphics
	■ Transfer of cutting values from technology database
	■ Use of DIN macros in the Teach-in program
	■ Conversion of Teach-in programs to smart.Turn programs (option 9)
Interactive contour programming (ICP) (option 8 or option 9)	■ Contour definition with linear and circular contour elements
	■ Immediate display of entered contour elements
	■ Calculation of missing coordinates, points of intersection, etc.
	■ Graphic display of all solutions for selection by the user if more than one solution is possible
	■ Chamfers, rounding arcs, and undercuts available as form elements
	■ Input of form elements immediately during contour creation or by superimposition later
	■ Changes to existing contours can be programmed
	■ Machining attributes available for individual contour elements
	■ Programming of the rear face for full-surface machining with the C and Y axes
C-axis machining on the front face and lateral surface (option 55)	■ Description of individual holes and hole patterns
	■ Description of figures and figure patterns for milling operations
	■ Creation of freely definable milling contours
Y-axis machining in the XY and ZY planes (option 70)	■ Description of individual holes and hole patterns
	■ Description of figures and figure patterns for milling operations
	■ Creation of freely definable milling contours

User functions

B-axis machining (option 54)	▪ Machining with the B axis
	▪ Tilting the working plane, rotating the machining position of the tool
	▪ **High Dynamic Turning** (abbreviated as: **HDT**) allows you to perform the most common turning operations with only one tool.
DXF import (option 42)	▪ Importing contours for turning
	▪ Importing contours for milling
smart.Turn programming (option 9)	▪ The basis is the "unit," which is the complete description of a machining block (geometry, technology and cycle data)
	▪ Dialog boxes divided into overview and detail forms
	▪ Fast navigation between the fillable forms and input groups via the smart.Turn keys
	▪ Context-sensitive help graphics
	▪ Start unit with global settings
	▪ Transfer of global values from the start unit
	▪ Transfer of cutting values from technology database
	▪ Units for all lathe and recessing operations
	▪ Use of the contours described with ICP for turning and recess turning operations
	▪ Units for all milling, boring, and drilling operations with the C axis (option 55)
	▪ Use of the patterns and contours described with ICP for C-axis operations (option 55)
	▪ Units for activating and deactivating the C axis (option 55)
	▪ Units for all boring, drilling, and milling operations with the Y axis (option 70)
	▪ Use of the patterns and contours described with ICP for Y-axis operations (option 70)
	▪ Special units for subprograms and program part repeats
	▪ Control graphics for the workpiece blank and finished part, as well as for C-axis contours (option 55) and Y-axis contours (option 70)
	▪ Turret assignment, magazine assignment and other setup information in the smart.Turn program
	▪ Parallel programming
	▪ Parallel simulation

User functions

DIN PLUS programming	■ Programming in DIN 66025 format
	■ Extended command format (**IF… THEN … ELSE…**)
	■ Simplified geometry programming (calculation of missing data)
	■ Powerful fixed cycles for turning, recessing, recess turning and thread machining
	■ Powerful fixed cycles for boring, drilling, and milling with the C axis (option 55)
	■ Powerful fixed cycles for boring, drilling, and milling with the Y axis (option 70)
	■ Subprograms
	■ Programming with variables
	■ Contour description with ICP (option 8 or option 9)
	■ Program verification graphics for workpiece blank and finished part
	■ Turret assignment, magazine assignment, and other setup information in the DIN PLUS program
	■ Conversion of smart.Turn units into DIN PLUS command sequences (option 9)
	■ Parallel programming
	■ Parallel simulation
Test graphics	■ Graphic simulation of the teach-in cycle run, the teach-in, smart.Turn, or DIN PLUS program
	■ Display of the tool paths as pencil-trace or cutting-path graphics, special identification of the rapid traverse paths
	■ Motion simulation (2-D material-removal graphic)
	■ Display of programmed contours
	■ Side or front face view, or 2-D view of (unrolled) cylindrical surface for verification of C-axis machining (option 55)
	■ View of front face (XY plane) and YZ plane for verification of Y-axis operations (option 70)
	■ Shifting and magnifying functions
	■ 3-D graphics for display of workpiece blank and finished part as solid-model view
Machining time analysis	■ Calculation of machining times and idle times
	■ Consideration of switching commands triggered by the CNC
	■ Representation of time per individual cycle or per tool change
TURN PLUS (option 63)	■ Automatic generation of smart.Turn programs
	■ Automatic cutting limitation through the definition of chucking equipment
	■ Automatic tool selection and turret assignment/magazine assignment

User functions

Tool database	■ For 250 tools
	■ For 999 tools (option 10)
	■ Tool description can be entered for every tool
	■ Automatic inspection of tool-tip position with respect to the contour
	■ Compensation of tool-tip position in the X/Y/Z plane
	■ High-precision correction via handwheel, capturing compensation values in the tool table
	■ Automatic tool-tip and cutter radius compensation
	■ Tool monitoring for lifetime of the insert (tool tip) or the number of workpieces produced
	■ Tool monitoring with automatic tool change in case of worn-out indexable insert (option 10)
	■ Management of multipoint tools (multiple inserts or reference points)
	■ Support for quick-change tool systems
Technology database (option 8 or option 9)	■ Access to cutting data after definition of workpiece material, cutting material and machining mode. The control distinguishes between 16 machining modes. Each workpiece-material/tool-material combination includes the cutting speed, the main and secondary feed rates, and the infeed for 16 machining modes.
	■ Automatic determination of the machining modes from the cycle or the machining unit
	■ The cutting data are entered in the cycle or in the unit as default values.
	■ 9 workpiece-material/tool-material combinations (144 entries)
	■ 62 workpiece-material/tool-material combinations (992 entries) (option 10)
User administration	Configurable linkage of permissions to user roles

User functions

Conversational languages	■ **ENGLISH** ■ **GERMAN** ■ **CZECH** ■ **FRENCH** ■ **ITALIAN** ■ **SPANISH** ■ **PORTUGUESE** ■ **SWEDISH** ■ **DANISH** ■ **FINNISH** ■ **DUTCH** ■ **POLISH** ■ **HUNGARIAN** ■ **RUSSIAN** ■ **CHINESE** ■ **CHINESE_TRAD** ■ **SLOVENIAN** ■ **KOREAN** ■ **NORWEGIAN** ■ **ROMANIAN** ■ **SLOVAK** ■ **TURKISH**

Accessories

Electronic handwheels	■ HR#180 panel-mounted handwheel with connection to position inputs ■ HR 130 panel-mounted handwheel ■ HR 510, HR 510 FS portable handwheel ■ HR 520, HR 520 FS portable handwheel with display ■ HR 550 FS portable wireless handwheel with display
Touch probe	Workpiece touch probes: ■ TS 150 touch trigger probe with cable connection ■ TS 260 touch trigger probe with cable connection ■ TS 750 touch trigger probe with cable connection ■ TS 740 touch trigger probe with infrared transmission ■ TT 460 touch trigger probe with radio or infrared transmission Tool touch probes: ■ TT 160 touch trigger probe ■ TT 460 touch trigger probe with radio or infrared transmission
DataPilot CP 640, MP 620	Control software for PCs for programming, archiving, and training for the MANUALplus620 control: ■ Full version with license for single station or multiple stations ■ Demo version (free of charge)

13.4 Compatibility in DIN programs

The format of DIN programs of the CNC PILOT 4290 predecessor control differs from the format of the MANUALplus 620. However, you can use the program converter to adapt programs of the predecessor control to the new control.

When opening an NC program, the control recognizes the programs of the predecessor control. The program concerned will be converted after a confirmation prompt. **CONV_...** will be prefixed to the program name. The program converter is also part of **Transfer** submode (**Organization** operating mode).

When converting DIN programs, it is also important to keep in mind that the predecessor controls use different solutions for tool management, parameter management, programming with variables and PLC programming.

Remember the following when converting DIN programs of the CNC PILOT 4290:

Tool call (**T** commands of the **TURRET** section):

- **T** commands containing a reference to the tool database are left unchanged (example: **T1 ID"342-300.1"**)
- **T** commands containing tool data cannot be converted

Variable programming:

- **D** variables (# variables) are replaced by # variables of the new syntax. Depending on the range of numbers, **#c** or **#l** or **#n** oder **#i** variables will be used.
- Special conversion rules: **#0** is converted to **#c30**, **#30** is converted to **#c51**
- **V** variables are replaced by #g variables. Braces used in assignments are removed. Braces used in expressions are changed to parentheses.
- Variable accesses to tool data, machine dimensions, **D** compensation values, parameter data, and events cannot be converted. These program sequences have to be adapted. Exception: The **Start blck search active E90[1]** event is converted to **#i6**
- Remember that – unlike the CNC PILOT 4290 – the interpreter of the control re-evaluates the lines every time the program is executed.

M functions:

- **M30** with **NS..** is converted to **M0 M99 NS**
- **M97** is removed for single-channel controls
- All other **M** functions are left unchanged

G functions:

- Currently, the control does not support the following **G** functions: **G98**, **G204**, **G710**, **G906**, **G907**, **G915**, **G918**, **G975**
- The following **G** functions are not supported by single-channel controls: **G62**, **G63**, **G162**
- A warning will be issued if the following **G** functions are used in a contour description: **G10**, **G38**, **G39**, **G52**, **G95**, **G149**. These functions are now modal
- Warnings may occur for the thread functions **G31**, **G32**, **G33**; it is recommended to test these functions
- The **Contour mirroring/shifting G121** function is converted to **G99** but the principle of function is compatible
- A warning will be issued for **G48** because the principle of function has changed.
- A warning will be issued for **G916**, **G917** and **G930** because the principle of function has changed. Functions must be supported by the PLC

Names of external subprograms:

- When an external subprogram is called, the converter prefixes **CONV_...** to the name

Multichannel programs on single-channel controls:

- On single-channel controls, programs for two slides are converted to a single slide and the Z-axis movements of the second slide are converted to **G1 W...** or **G701 W...**
  - In the program head, **#SLIDE $1$2** is replaced by **#SLIDE $1**
  - $ statements preceding the block number are removed
  - **$2 G1 Z...** is converted to **G1 W...** and **G701 Z...** to **G701 W...**
  - The word **SLIDE ALLOCATION** is removed (but remembered internally for converting the subsequent blocks)
  - Synchronization statements **$1$2 M97** are removed
  - Datum shifts for slide 2 are saved as comments; paths of traverse are provided with a warning.

Nonconvertible elements:

- If the ISO program contains nonconvertible elements, the corresponding NC block is saved as a comment. The word **Warning** is inserted in front of this comment. Depending on the situation, the nonconvertible command is taken into the comment line, or the nonconvertible NC block follows the comment.

NOTICE

Danger of collision!

Converted NC programs may have incorrectly converted contents (machine-dependent) or nonconvertible contents. Danger of collision during machining!

▶ Adapt converted NC programs to the current control
▶ Test the NC program in **Simulation** submode, using the graphic displayed there

---

13.5 Syntax elements provided by the control

Meaning of the symbols used in the table:

✓	Compatible behavior; if necessary, the program converter converts the functions so that they are compatible with the control.
X	Behavior changed; the program has to be tested in the individual case
–	Function is not available or is replaced by different functionality
★	This function is available for controls with multichannel capability.
◆	This function is planned for future software versions and will initially be required for controls with multichannel capability only

Section codes		
Program head	**PROGRAM HEAD**	✓
	PLATE MAGAZINE	✓
	TURRET	✓
	MAGAZINE	✓
	CHUCKING EQUIPMENT	X
Contour definition	**CONTOUR** / **Contour group**	X
	BLANK	✓
	FINISHED PART	✓
	AUXILIARY CONTOUR	✓
C-axis contours	**FRONT**	✓
	REAR SIDE	✓
	LATERAL	✓
Workpiece machining	**MACHINING**	✓
	SLIDE ALLOCATION	★
	END	✓
Subprograms	**SUBROUTINE**	✓
	Return	✓
Others	**CONST**	✓
Y-axis contours	**FRONT_Y**	✓
	REAR SIDE_Y	✓
	LATERAL_Y	✓

G codes for turning contours

Workpiece-blank definition	**G20-Geo** Chuck part, cylinder/tube	✓
	G21-Geo Cast part	✓
Basic contour elements	**G0-Geo** Starting point of contour	✓
	G1-Geo Line segment	✓
	G2-Geo Circular arc with incremental center dimensioning	✓
	G3-Geo Circular arc with incremental center dimensioning	✓
	G12-Geo Circular arc with absolute center dimensioning	✓
	G13-Geo Circular arc with absolute center dimensioning	✓
Contour form elements	**G22-Geo** Recess (standard)	✓
	G23-Geo Recess/relief turn	✓
	G24-Geo Thread with undercut	✓
	G25-Geo Undercut contour	✓
	G34-Geo Thread (standard)	✓
	G37-Geo Thread (general)	✓
	G49-Geo Bore hole at turning center	✓
Help commands for contour definition	**G7-Geo** Precision stop on	✓
	G8-Geo Precision stop off	✓
	G9-Geo Precision stop blockwise	✓
	G10-Geo Peak-to-valley height	X
	G38-Geo Feed rate reduction	X
	G39-Geo Attributes for superimposed elements	–
	G52-Geo Blockwise oversize	X
	G95-Geo Feed per revolution	X
	G149-Geo Additive compensation	X

G codes for C-axis contours

Overlapping contours	**G308-Geo** Start of pocket/island	✓
	G309-Geo End of pocket/island	✓
Front and rear face contours	**G100-Geo** Starting point of face contour	✓
	G101-Geo Line segment on front face	✓
	G102-Geo Circular arc on front face	✓
	G103-Geo Circular arc on front face	✓
	G300-Geo Bore hole on front face	✓
	G301-Geo Linear slot on front face	✓
	G302-Geo Circular slot on front face	✓
	G303-Geo Circular slot on front face	✓
	G304-Geo Full circle on front face	✓
	G305-Geo Rectangle on front face	✓
	G307-Geo Eccentric polygon on front face	✓
	G401-Geo Linear pattern on front face	✓
	G402-Geo Circular pattern on front face	✓
Lateral surface contours	**G110-Geo** Starting point of lateral surface contour	✓
	G111-Geo Line segment on lateral surface	✓
	G112-Geo Circular arc on lateral surface	✓
	G113-Geo Circular arc on lateral surface	✓
	G310-Geo Bore hole on lateral surface	✓
	G311-Geo Linear slot on lateral surface	✓
	G312-Geo Circular slot on lateral surface	✓
	G313-Geo Circular slot on lateral surface	✓
	G314-Geo Full circle on lateral surface	✓
	G315-Geo Rectangle on lateral surface	✓
	G317-Geo Eccentric polygon on lateral surface	✓
	G411-Geo Linear pattern on lateral surface	✓
	G412-Geo Circular pattern on lateral surface	✓

G codes for Y-axis contours

XY plane	**G170-Geo** Starting point of contour	✓
	G171-Geo Line segment	✓
	G172-Geo Circular arc	✓
	G173-Geo Circular arc	✓
	G370-Geo Hole	✓
	G371-Geo Linear slot	✓
	G372-Geo Circular slot	✓
	G373-Geo Circular slot	✓
	G374-Geo Full circle	✓
	G375-Geo Rectangle	✓
	G376-Geo Single surface	✓
	G377-Geo Eccentric polygon	✓
	G471-Geo Linear pattern	✓
	G472-Geo Circular pattern	✓
	G477-Geo Centric polygon	✓
YZ plane	**G180-Geo** Starting point of contour	✓
	G181-Geo Line segment	✓
	G182-Geo Circular arc	✓
	G183-Geo Circular arc	✓
	G380-Geo Hole	✓
	G381-Geo Linear slot	✓
	G382-Geo Circular slot	✓
	G383-Geo Circular slot	✓
	G384-Geo Full circle	✓
	G385-Geo Rectangle	✓
	G387-Geo Eccentric polygon on lateral surface	✓
	G481-Geo Linear pattern	✓
	G482-Geo Circular pattern	✓
	G386-Geo Single surface	✓
	G487-Geo Centric polygon	✓

G codes for machining

Tool positioning without machining	**G0** Positioning at rapid traverse	✓
	G14 Move to the tool change position	✓
	G701 Rapid traverse to machine coordinates	✓
Simple linear and circular movements	**G1** Linear movement	✓
	G2 Circular movement with incremental center dimensioning	✓
	G3 Circular movement with incremental center dimensioning	✓
	G12 Circular movement with absolute center dimensioning	✓
	G13 Circular movement with absolute center dimensioning	✓
Feed rate and spindle speed	**Gx26** Speed limitation	✓
	G48 Reduce rapid traverse	X
	G64 Interrupted feed	✓
	G192 Feed per minute for rotary axis	–
	Gx93 Feed per tooth	✓
	G94 Feed per minute	✓
	Gx95 Feed per revolution	✓
	Gx96 Constant surface speed	✓
	Gx97 Speed	✓
Tool-tip radius compensation	**G40** Switch off TRC/MCRC	✓
	G41 TRC/MCRC, left	✓
	G42 TRC/MCRC, right	✓
Datum shifts	**G51** Relative datum shift	✓
	G53 Parameter-dependent datum shift	✓
	G54 Parameter-dependent datum shift	✓
	G55 Parameter-dependent datum shift	✓
	G56 Additive datum shift	✓
	G59 Absolute datum shift	✓
	G121 Contour mirroring/shifting	✓
	G152 Datum shift, C axis	✓
	G920 Deactivate datum shift	✓
	G921 Deactivate datum shift, tool dimensions	✓
	G980 Activate datum shift	✓
	G981 Activate datum shift, tool dimensions	✓

G codes for machining

Oversizes	**G50** Switch off oversize	✓
	G52 Switch off oversize	✓
	G57 Axis-parallel oversize	✓
	G58 Contour-parallel oversize	✓
Safety clearances	**G47** Set safety clearances	✓
	G147 Safety clearance (milling cycles)	✓
Tools, types of compensation	**T** Tool call	✓
	G148 Switching the tool edge compensation	✓
	G149 Additive compensation	✓
	G150 Compensation of right-hand tool tip	✓
	G151 Compensation of left-hand tool tip	✓
	G710 Adding tool dimensions	◆

Cycles for turning

Simple turning cycles	**G80** End of cycle	✓
	G81 Simple longitudinal roughing	✓
	G82 Simple face roughing	✓
	G83 Simple contour repeat cycle	✓
	G85 Undercut	✓
	G86 Simple recessing cycle	✓
	G87 Transition radii	✓
	G88 Chamfers	✓
Drilling cycles	**G36** Tapping	✓
	G71 Simple drilling cycle	✓
	G72 Counterboring, countersinking, etc.	✓
	G73 Tapping cycle	✓
	G74 Deep hole drilling cycle	✓
Contour-based turning cycles	**G810** Longitudinal roughing cycle	✓
	G820 Face roughing cycle	✓
	G830 Contour-parallel roughing cycle	✓
	G835 Contour-parallel with neutral tool	✓
	G860 Universal recessing cycle	✓
	G866 Simple recessing cycle	✓
	G869 Recess turning cycle	✓
	G890 Finishing cycle	✓
Thread cycles	**G31** Thread cycle	✓
	G32 Single thread cycle	✓
	G33 Single thread cut (Thread single path)	✓
	G933 Thread switch	–
	G799 Thread milling, axial	✓
	G800 Thread milling in XY plane	✓
	G806 Thread milling in YZ plane	✓

Synchronization codes

Assigning contour to operation	**G98** Assignment of spindle to workpiece	–
	G99 Workpiece group	X
Slide synchronization	**G62** One-sided synchronization	★
	G63 Synchronous start of slides	★
	G162 Synchronization marking	★
Contour follow-up	**G702** Storing/loading Contour follow-up	✓
	G703 Contour follow-up ON/OFF	✓
	G706 K default branch	–
Spindle synchronization, workpiece transfer	**G30** Converting and mirroring	✓
	G121 Contour mirroring/shifting	✓
	G720 Spindle synchronization	✓
	G905 Measuring C-angle offset	–
	G906 Measuring angular offset during spindle synchronization	–
	G916 Traversing to a fixed stop	✓
	G917 Controlled parting using lag error monitoring	✓
	G991 Controlled parting using spindle monitoring	–
	G992 Values for controlled parting	–

C-axis machining

C axis	**G119** No. of C axis	✓
	G120 Reference diameter for lateral-surface machining	✓
	G152 Datum shift, C axis	✓
	G153 Standardize C axis	✓
Front/rear-face machining	**G100** Rapid traverse, front face	✓
	G101 Synchronous start of slides	✓
	G102 Circular arc on front face	✓
	G103 Circular arc on front face	✓
Milling cycles	**G799** Thread milling, axial	✓
	G801 Engraving on front face	✓
	G802 Engraving on lateral surface	✓
	G840 Contour milling	✓
	G845 Pocket milling, roughing	✓
	G846 Pocket milling, finishing	✓
Lateral-surface machining	**G110** Rapid traverse, lateral surface	✓
	G111 Linear path on lateral surface	✓
	G112 Circular arc on lateral surface	✓
	G113 Circular arc on lateral surface	✓

Variable programming, program branches

Programming with variables	**# variables** Evaluation during program conversion	✓
	V variables Evaluation during program run	✓
Program branches, program repeats	**IF..THEN..** Program branching	✓
	WHILE.. Program repeat	✓
	SWITCH.. Program branching	✓
Special functions	**$** Slide code	✓
	/ Skip level	✓
Data input and data output	**INPUT** Input (# variables)	✓
	WINDOW Open output window (# variables)	✓
	PRINT Output (# variables)	✓
	INPUTA Input (V variables)	✓
	WINDOWA Open output window (V variables)	✓
	PRINTA Output (V variables)	✓
Subprograms	**L** Subprogram call	✓

Measuring functions, load monitoring

In-process measrmnt.	**G910** Activate In-process measrmnt.	✓
	G912 Actual value capture for In-process measrmnt.	✓
	G913 Deactivate In-process measrmnt.	✓
	G914 Switch off probe monitoring	✓
Post-process measuring	**G915** Post-process measuring	◆
Load monitoring	**G995** Specifying the monitoring zone	✓
	G996 Type of load monitoring	✓

Other G codes

Other G codes		
	G4 Dwell time	✓
	G7 Precision stop on	✓
	G8 Precision stop off	✓
	G9 Precision stop (blockwise)	✓
	G15 Move rotary axes	–
	G60 Deactivate protection zone	✓
	G65 Display chucking equipment	✓
	G66 Component position	◆
	G204 Waiting for time	◆
	G717 Updating nominal values	–
	G718 Move lag error	–
	G901 Actual values in variables	✓
	G902 Datum shift in variables	✓
	G903 Lag error in variables	✓
	G907 Block speed monitoring off	◆
	G908 Feed rate override 100 %	✓
	G909 Interpreter stop	✓
	G918 Feedforward control	–
	G919 Spindle override 100 %	✓
	G920 Deactivate datum shift	✓
	G921 Deactivate datum shifts, tool lengths	✓
	G922 Spindle speed at constant V	–
	G930 Sleeve monitoring	✓
	G940 Internal T number	–
	G941 Transfer pocket compensation values	–
	G975 Servo lag limit	◆
	G980 Activate datum shift	✓
	G981 Activate datum shifts, tool lengths	✓

B-axis and Y-axis machining

Working planes	**G16** Tilting working plane	✓
	G17 XY plane (front or rear face)	✓
	G18 XZ plane (turning)	✓
	G19 YZ plane (plan view / lateral surface)	✓
Tool positioning without machining	**G0** Positioning at rapid traverse	✓
	G14 Move to the tool change position	✓
	G600 Preselect tool	✓
	G701 Rapid traverse to machine coordinates	✓
	G714 Insert magazine tool	◆
	G712 Define tool position	◆
Milling cycles	**G841** Area milling, roughing	✓
	G842 Area milling, finishing	✓
	G843 Centric polygon milling, roughing	✓
	G844 Centric polygon milling, finishing	✓
	G845 Pocket milling, roughing	✓
	G846 Pocket milling, finishing	✓
	G800 Thread milling in XY plane	✓
	G806 Thread milling in YZ plane	✓
	G803 Engraving in XY plane	✓
	G804 Engraving in YZ plane	✓
	G808 Hobbing	✓
Simple linear and circular movements	**G1** Linear path	✓
	G2 Circular arc with incremental center coordinates	✓
	G3 Circular arc with incremental center coordinates	✓
	G12 Circular arc with absolute center coordinates	✓
	G13 Circular arc with absolute center coordinates	✓

14

Overview of Cycles

14.1 Workpiece blank cycles and single cut cycles

14.2 Turning cycles

Turning cycles			Page
		Overview	Page 220
		Cut longitudinal: Roughing and finishing cycle for simple contours	Page 222
		Cut transverse: Roughing and finishing cycle for simple contours	Page 224
		Cut with longitudinal plunge Roughing and finishing cycle for simple contours	Page 238
		Cut with transverse plunge: Roughing and finishing cycle for simple contours	Page 240
		ICP long. parallel contour: Roughing and finishing cycle for any type of contour	Page 254
		ICP transv. parallel contour: Roughing and finishing cycle for any type of contour	Page 257
		ICP cutting longitudinal: Roughing and finishing cycle for any type of contour	Page 264
		ICP cutting transverse: Roughing and finishing cycle for any type of contour	Page 266

14.3 Recessing and recess-turning cycles

Recessing cycles			Page
		Overview	Page 276
		Recessing radial: Recessing and finishing cycles for simple contours	Page 278
		Recessing axial: Recessing and finishing cycles for simple contours	Page 280
		Recessing radial ICP: Recessing and finishing cycles for any contour	Page 294
		Recessing axial ICP: Recessing and finishing cycles for any contour	Page 296
		Undercut H	Page 326
		Undercut K	Page 328
		Undercut U	Page 329
		Cut off: Cycle for parting the workpiece	Page 330

Recess-turning cycles			Page
		Overview	Page 301
		Radial recess turning: Recess-turning and finishing cycles for simple contours	Page 302
		Axial recess turning: Recess-turning and finishing cycles for simple contours	Page 304
		ICP-Recess turning long.: Recess-turning and finishing cycles for any type of contour	Page 318
		ICP-Recess turning transv.: Recess-turning and finishing cycles for any type of contour	Page 320

14.4 Thread cycles

14.5 Drilling cycles

Drilling cycles			Page
		Overview	Page 363
		Drilling axial: For drilling single holes and patterns	Page 363
		Drilling radial: For drilling single holes and patterns	Page 366
		Deep drilling axial: For drilling single holes and patterns	Page 368
		Deep drilling radial: For drilling single holes and patterns	Page 371
		Tapping axial: For drilling single holes and patterns	Page 374
		Tapping radial: For drilling single holes and patterns	Page 376
		Thread milling: For milling threads in existing holes	Page 378

14.6 Milling cycles

Milling cycles		Page
	Overview	Page 382
	Rap. trav. positioning: Activate C axis. Position tool and spindle	Page 383
	Axial slot: For milling single slots or slot patterns	Page 384
	Axial figure: For milling single figures	Page 388
	Axial contour ICP: For milling single ICP contours or contour patterns	Page 396
	Face milling: For milling surfaces or centric polygons	Page 404
	Radial slot: For milling single slots or slot patterns	Page 386
	Radial figure: For milling single figures	Page 392
	Radial contour ICP: For milling single ICP contours or contour patterns	Page 400
	Hel. slot milling, radial: For milling helical slots	Page 407
	Thread milling: for milling threads in existing holes	Page 378

Index

HEIDENHAIN