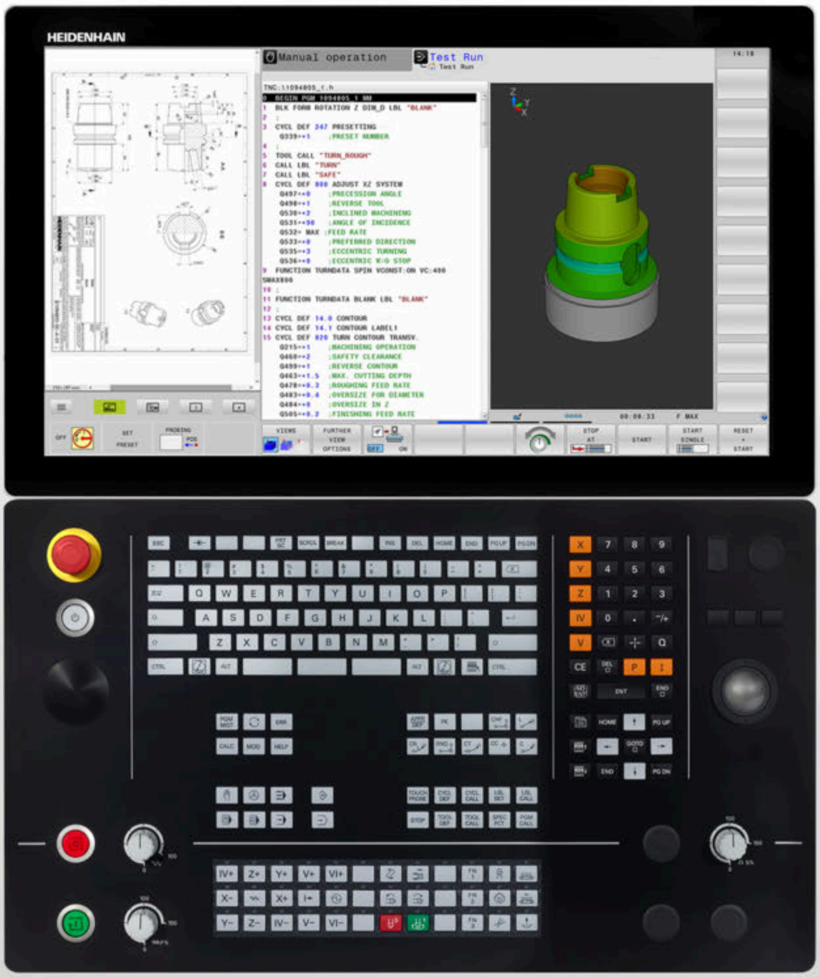
HEIDENHAIN



TNC 640

User's Manual
Conversational Programming

NC Software
340590-10
340591-10
340595-10

English (en)
10/2019

Controls and displays

Keys

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

Further information: "Operating the touchscreen", Page 545

Keys on the screen

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

Alphabetic keyboard

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

Machine operating modes

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

Programming modes

Key	Function
	Programming
	Test Run

Entering and editing coordinate axes and numbers

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

Tool functions

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

Managing NC programs and files, control functions

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

Navigation keys

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

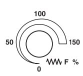
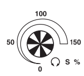
Cycles, subprograms and program section repeats

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

Program path contours

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

Potentiometer for feed rate and spindle speed

Feed rate	Spindle speed
	

Contents

1

Fundamentals

1.1 About this manual

Safety precautions

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

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

⚠ DANGER

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

⚠ WARNING

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

⚠ CAUTION

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

NOTICE

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

Sequence of information in precautionary statements

All precautionary statements comprise the following four sections:

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

Informational notes

Observe the informational notes provided in these instructions to ensure reliable and efficient operation of the software.
In these instructions, you will find the following informational notes:

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

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

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

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

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

tnc-userdoc@heidenhain.de

1.2 Control model, software and features

This manual describes programming functions provided by controls as of the following NC software numbers.

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

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

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

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

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

- Tool measurement with the TT

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

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

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

User's Manual for Setup, Testing and Running NC Programs:
All information for setting up the machine as well as for testing and running your NC programs is provided in the User's Manual for **Setup, Testing and Running NC Programs**. If you need this User's Manual, please contact HEIDENHAIN.
ID: 1261174-xx

Software options

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

Additional Axis (options 0 to 7)	
Additional axis	Additional control loops 1 to 8

Advanced Function Set 1 (option 8)	
Expanded functions Group 1	**Machining with rotary tables** ■ Cylindrical contours as if in two axes ■ Feed rate in distance per minute **Coordinate conversions:** Tilting the working plane

Advanced Function Set 2 (option 9)	
Expanded functions Group 2 Export license required	**3-D machining:** ■ 3-D tool compensation through surface-normal vectors ■ Using the electronic handwheel to change the angle of the swivel head during program run; the position of the tool point remains unchanged (TCPM = Tool Center Point Management) ■ Keeping the tool normal to the contour ■ Tool radius compensation normal to the tool direction ■ Manual traverse in the active tool-axis system **Interpolation:** Linear in > 4 axes (export license required)

HEIDENHAIN DNC (option 18)	
	Communication with external PC applications over COM component

Dynamic Collision Monitoring – DCM (option 40)	
Dynamic Collision Monitoring	■ The machine manufacturer defines objects to be monitored ■ Warning in Manual operation ■ Collision monitoring in the Test Run mode ■ Program interrupt in Automatic operation ■ Includes monitoring of 5-axis movements

CAD Import (option 42)	
CAD import	■ Support for DXF, STEP and IGES ■ Adoption of contours and point patterns ■ Simple and convenient specification of presets ■ Selecting graphical features of contour sections from conversational programs

Adaptive Feed Control – AFC (option 45)	
Adaptive Feed Control	**Milling:** ■ Recording the actual spindle power by means of a teach-in cut ■ Defining the limits of automatic feed rate control ■ Fully automatic feed control during program run **Turning (option 50):** ■ Cutting force monitoring during machining

KinematicsOpt (option 48)	
Optimizing the machine kinematics	■ Backup/restore active kinematics ■ Test active kinematics ■ Optimize active kinematics

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

KinematicsComp (option 52)	
Three-dimensional compensation	Compensation of position and component errors

OPC UA NC Server 1 - 6 (Options 56 - 61)	
Standardized interface	The OPC UA NC Server provides a standardized interface (OPC UA) for external access to the control's data and functions These software options allow you to create up to six parallel client connections

3D-ToolComp (option 92)	
3-D tool radius compensation depending on the tool's contact angle Export license required	■ Compensate the deviation of the tool radius depending on the tool's contact angle ■ Compensation values in a separate compensation value table ■ Prerequisite: Working with surface normal vectors (**LN** blocks)

Extended Tool Management (option 93)	
Extended tool management	Python-based

Advanced Spindle Interpolation (option 96)	
Interpolating spindle	**Interpolation turning:** ■ Cycle 291: Interpolation turning, coupling ■ Cycle 292: Interpolation turning, contour finishing

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

Remote Desktop Manager (option 133)	
Remote operation of external computer units	■ Windows on a separate computer unit ■ Incorporated in the control's interface
Synchronizing Functions (option 135)	
Synchronization functions	**Real Time Coupling – RTC:** Coupling of axes
Visual Setup Control – VSC (option 136)	
Camera-based monitoring of the setup situation	■ Record the setup situation with a HEIDENHAIN camera system ■ Visual comparison of planned and actual status in the workspace
State Reporting Interface – SRI (option 137)	
HTTP accesses to the control status	■ Reading out the times of status changes ■ Reading out the active NC programs
Cross Talk Compensation – CTC (option 141)	
Compensation of axis couplings	■ Determination of dynamically caused position deviation through axis acceleration ■ Compensation of the TCP (**T**ool **C**enter **P**oint)
Position Adaptive Control – PAC (option 142)	
Adaptive position control	■ Changing of the control parameters depending on the position of the axes in the working space ■ Changing of the control parameters depending on the speed or acceleration of an axis
Load Adaptive Control – LAC (option 143)	
Adaptive load control	■ Automatic determination of workpiece weight and frictional forces ■ Changing of control parameters depending on the actual mass of the workpiece
Active Chatter Control – ACC (option 145)	
Active chatter control	Fully automatic function for chatter control during machining
Active Vibration Damping – AVD (option 46)	
Active vibration damping	Damping of machine oscillations to improve the workpiece surface
Batch Process Manager (option 154)	
Batch process manager	Planning of production orders
Component Monitoring (option 155)	
Component monitoring without external sensors	Monitoring configured machine components for overload

Grinding (Option 156)	
Jig grinding	■ Reciprocating stroke cycles ■ Cycles for dressing ■ Support of the "dressing tool" and "grinding tool" tool types
Gear Cutting (option 157)	
Machining gear systems	■ Cycle 285: Define gear wheel ■ Cycle 286: Gear hobbing ■ Cycle 287: Gear skiving
Advanced Function Set Turning (option 158)	
Advanced turning functions	Cycle 883: Simultaneous turning
Opt. contour milling (Option 167)	
Optimized contour cycles	■ Cycle 271: **OCM CONTOUR DATA** ■ Cycle 272: **OCM ROUGHING** ■ Cycle 273: **OCM FINISHING FLOOR** ■ Cycle 274: **OCM FINISHING SIDE**

Feature Content Level (upgrade functions)

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

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

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

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

Intended place of operation

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

Legal information

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

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

New functions 34059x-09

- It is now possible to work with cutting data tables, see "Working with cutting data tables", Page 203
- The **TCPM** function can also consider spatial angles for Peripheral Milling, see "Peripheral milling: 3-D radius compensation with TCPM and radius compensation (RL/RR)", Page 456
- New **PLANE XY ZX YZ** soft key for selecting the working plane during FK programming, see "Fundamentals", Page 173
- In **Test Run** operating mode, a counter defined in the NC program is simulated, see "Defining a counter", Page 381
- An NC program you called can be edited when it has been completely executed in the calling NC program.
- In the CAD viewer, you can define the preset or the datum by directly entering the values in the list view window, see "Data Transfer from CAD Files", Page 467
- With **TOOL DEF**, you can use QS parameters for entering the data, see "Entering tool data into the NC program", Page 126
- You can now use QS parameters to read from and write to freely definable tables, see "FN 27: TABWRITE – Write to a freely definable table", Page 390
- The FN 16 function was extended to include the ***** input character that can be used to write comment lines, see "Creating a text file", Page 285
- New output format for the FN 16 function **%RS** that you can use to output texts without formatting, see "Creating a text file", Page 285
- The FN18 functions have been expanded, see "FN 18: SYSREAD – Reading system data", Page 293

Further information: User's Manual for **Setup, Testing and Running NC Programs**

- The new user administration enables you to create and administrate users with different access rights.
- The new **Component Monitoring** software option enables automatic checking of defined machine components for overload.
- With the new HOST COMPUTER MODE function, you can transfer command to an external host computer.
- With the **State Reporting Interface** (**SRI**), HEIDENHAIN provides a simple and reliable interface for acquiring the operating states of your machine.
- The basic rotation is taken into account in the **Manual Operation** mode.
- The new **PROGRAM + MACHINE** screen layout shows you the NC program, collision objects and the workpiece.
- The new **MACHINE** screen layout shows you the collision objects and the workpiece.
- The screen layout soft keys were adapted.
- The additional status display shows the path and angle tolerances without Cycle 32 being active.
- The additional status display indicates whether the path and angle tolerances are limited by DCM.

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

Modified functions 34059x-09

- The **PLANE** functions provide the alternative **SYM**selection option in addition to **SEQ**, see "Selection of tilting possibilities SYM (SEQ) +/–", Page 426
- The cutting data calculator has been improved, see "Cutting data calculator", Page 201
- The **CAD-Viewer** now outputs **PLANE SPATIAL** instead of **PLANE VECTOR**, see "Defining the datum", Page 477
- The **CAD-Viewer** now outputs 2-D contours by default.
- When programming straight-line blocks, the **&Z** option is no longer shown by default, see "FUNCTION PARAXMODE", Page 368
- The control does not run a tool change macro if neither a tool name nor a tool number is programmed in the tool call, but the same tool axis as in the previous **TOOL CALL** block, see "Calling the tool data", Page 127
- The control issues an error message if you combine an FK block with M89.

- When using **SQL UPDATE** and **SQL INSERT**, the control checks the length of the table columns to be written to, see "SQL UPDATE", Page 310, see "SQL INSERT", Page 312
- When using the FN16 function, M_CLOSE and M_TRUNCATE have the same effect as far as output to the screen is concerned, see "Displaying messages on the control screen", Page 292

Further information: User's Manual for **Setup, Testing and Running NC Programs**

- The **Batch Process Manager** can now be opened in the **Programming**, **Program run, full sequence** and **Program run, single block** operating modes.
- In the **Test Run** operating mode, the **GOTO** key now has the same effect as in the other operating modes.
- If axis angle not equal to tilt angle, the control no longer issues an error message during presetting with manual probing functions, but opens the **Working plane is inconsistent** menu.
- The **ACTIVATE PRESET** soft key also updates the values of a line activated in the preset management.
- From the third desktop you can switch to any operating mode using the operating mode keys.
- The additional status display in the **Test Run** operating mode was adapted to match that of the **Manual operation** mode.
- The control allows updating of the web browser.
- The Remote Desktop Manager allows you to enter an additional waiting time for the shutdown connection.
- The obsolete tool types were removed from the tool table. The **Undefined** type is assigned to any existing tools of these tool types.
- In the expanded tool management, you can now go to the context-sensitive on-line help even while editing the tool form.
- The screensaver glideshow was removed.
- The machine tool builder can specify the axis-specific effect of a shift (mW-CS) of the rotary axes.
- The machine tool builder can define the minimum distance between two collision-monitored objects in the **Manual operation** mode.
- The machine tool builder can specify which M functions to allow in the **Manual Operation** mode.
- The machine tool builder can define the default values for the L-OFFS and R-OFFS columns in the tool table.

New and modified cycle functions 34059x-09

Further information:Cycle Programming User's Manual

- New Cycle 285 DEFINE GEAR WHEEL (option 157).
- New Cycle 286 GEAR HOBBING (option 157).
- New Cycle 287 GEAR SKIVING (option 157).
- New Cycle 883 TURNING SIMULTANEOUS FINISHING (Option 50 and Option 158).
- New Cycle 1410 PROBING ON EDGE.
- New Cycle 1411 PROBING TWO CIRCLES.
- New Cycle 1420 PROBING IN PLANE.
- Automatic Touch Probe Cycles 408 to 419 take chkTiltingAxes (no. 204600) into account during presetting.
- Touch Probe Cycles 41x, automatic preset measurement: New behavior of cycle parameters Q303 MEAS. VALUE TRANSFER and Q305 NUMBER IN TABLE.
- In Cycle 420 MEASURE ANGLE, the data from the cycle and the touch probe table is taken into account during pre-positioning.
- Cycle 444 PROBING IN 3-D checks whether the positions of the rotary axes agree with the tilt angles depending on the setting of the optional machine parameter.
- The help graphic in Cycle 444 PROBING IN 3-D for Q309 ERROR REACTION has been modified and this cycle takes into account a TCPM.
- Cycle 450 SAVE KINEMATICS does not write the same values during restoring.
- Cycle 451 MEASURE KINEMATICS was expanded to include value 3 in cycle parameter Q406 MODE.
- In Cycles 451 MEASURE KINEMATICS and 453 KINEMATICS GRID, the radius of the calibration sphere is only monitored during the second measurement.
- A simulated touch probe is considered in the simulation. The simulation runs without error message.
- The REACTION column was added to the touch probe table.
- In Cycle 24 SIDE FINISHING, a tangential helix is used for approaching and departing in the last infeed.
- Parameter Q367 SURFACE POSITION was added to Cycle 233 FACE MILLING.
- Cycle 257 CIRCULAR STUD, now uses Q207 FEED RATE MILLING for roughing, too.
- The configuration CfgGeoCycle (no. 201000) is taken into account in Cycles 291 COUPLG.TURNG.INTERP. and 292 CONTOUR.TURNG.INTRP..
- Parameter Q531 ANGLE OF INCIDENCE was extended to 0.001° in Cycle 800 ADJUST XZ SYSTEM.
- Machine parameter CfgThreadSpindle (no. 113600) is available for use.

New functions 34059x-10

- The jig grinding function (Option 156) allows to machine a workpiece using a grinding tool. On the path contour, a superimposed reciprocating stroke is possible, see "Grinding operations on milling machines (option 156)", Page 538
- The **FUNCTION DRESS** dressing function (Option 156) allows to dress grinding tools, see "Dressing (option 156)", Page 541
- Using the compensation tables, the control can compensate for deviations in the tool coordinate system (T-CS) or working plane coordinate system (WPL-CS), also during the program run, see "Compensation table", Page 377
- In the **Batch Process Manager**, the common collision check of all NC programs used for a pallet is available, see "Opening the Batch Process Manager ", Page 501
- The column order of a table created with the **CREATE TABLE** function corresponds to the order within the **AS SELECT** command, see "SQL EXECUTE", Page 304
- With the **FUNCTION TCPM** function, you can define a feed-rate limit of the compensation movements, see "FUNCTION TCPM (option 9)", Page 443
- The **FUNCTION TCPM** function is available for ISO programming, see "FUNCTION TCPM (option 9)", Page 443
- The control performs a backup of active NC programs to a service file up to a maximum size of 10 MB.
- The FN18 functions have been extended, see "FN 18: SYSREAD – Reading system data", Page 293
- The machine tool builder uses an optional machine parameter to define the distance to a software limit switch or a collision object for retraction movements.
- The machine tool builder defines in an optional machine parameter whether the control will automatically clear pending warning or error messages when a new NC program is selected or the previous NC program is restarted, see "Clearing errors", Page 212

Further information: User's Manual for **Setup, Testing and Running NC Programs**

- The **OPC UA NC Server** software options 1 - 6 (Options 51 - 56) provide a standardized interface (OPC UA) for remote access to the control's data and functions.
- To simplify the setup of an OPC UA application, the control provides a configuration assistant as a HEROS function.
- In the default scope of delivery, the control provides high resolution of the display steps without the **Display Step** (Option 23) software option.
- Additional tool types are available for the definition of grinding tools and dressing tools.
- The **TOOL** tab in the additional status display shows specific data for grinding tools and dressing tools.
- Extended Tool Management also allows to apply the current position value as the tool length.
- The general status display shows an active tool radius compensation using various icons.

- With the **ACTIVATE AUTOMATIC SAVING** soft key, it is possible to define an error number whose occurrence automatically causes the creation of a service file.
- You can take over the position values, axis by axis, to a datum table in the **Program Run Single Block** and **Program Run Full Sequence** operating modes.
- The **POS HR** tab of the additional status display indicates whether the defined Max. val. from **M118** or from the **Global Program Settings** function is applied.
- In the **BLANK IN WORK SPACE** function, the **RESET REFERENCE POINT** soft key sets the principal axis values of the current preset to 0.
- In the **BLANK IN WORK SPACE** function, the **Load machine status** soft key is available.
- The control uses the current preset of the **Test Run** operating mode for simulation.
- The **ADOPT** menu displays either the defined axis angles or the spatial angles.
- While manual probing functions are active, the control temporarily disables the **Global Program Settings** function.
- In the **Global settings** function, the **ACTIVATE GLOBAL SETTINGS** soft key allows you to restore the settings that were last active.

  In the file manager, the **ADVANCED ACCESS RIGHTS** soft key allows you to assign file-specific access rights.

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

Functions changed 34059x-10

- The control includes the QR parameters in the backup, see "Principle and overview of functions", Page 264
- With the **SQL EXECUTE** and **SQL SELECT** SQL commands, it is possible to use composite QS parameters, see "SQL EXECUTE", Page 304
- Any display filter you have set in the file manager will remain effective even after a control restart, see "Selecting drives, directories and files", Page 108
- In addition to the **FN 9** step function, it is possible to use the **FN 10** function, which performs a comparison, for QS parameters and texts, see "Programming if-then decisions", Page 277

- The control executes the **FN 27: TABWRITE** function and **FUNCTION FILE** in the **Program run, single block** and **Program run, full sequence** operating modes only.
- In machine parameters **fn16DefaultPath** (no. 102202) and **fn16DefaultPathSim** (no. 102203), you can define the path for the output of the **FN 16** function, see "FN 16: F-PRINT – Formatted output of text and Q parameter values", Page 285

Further information: User's Manual for **Setup, Testing and Running NC Programs**

- In tool management, the control only displays the input fields related to the selected tool type.
- In the turning tool table, the default value for the **CUTLENGTH** column is 0.
- In the preset table, the input range of columns **SPA**, **SPB**, **SPC**, **A_OFFS**, **B_OFFS**, and **C_OFFS** was extended to +/- 99999.99999.
- On 19" and 24" screens, the control displays up to 10 axes in the additional status display,.
- Besides other information, the measuring function of the **Test Run** operating mode displays information on the tool.
- If user administration is active, the **Retraction after power interruption** function requires the **NC.OPModeManual**.
- If user administration is active, the **Global Program Settings** function requires the **NC.OPModeMDI**.
- In the additional status display, the **CM** and **CM Detail** tabs are replaced with the **MON** and **MON Detail** tabs.
- When capturing the **Program run:** machine times, the control only considers the active machining status. It is shown by the green **NC Start** icon in the status display.
- Remote access is identified by a new icon.
- On handwheels that have a display, the smallest speed level that can be set is 1/1000 of the maximum handwheel speed.

New and changed cycle functions 34059x-10

Further information:Cycle Programming User's Manual

- New point pattern cycle 224 DATAMATRIX CODE PATTERN for the creation of a DataMatrix code.
- New cycle 238 MEASURE MACHINE STATUS for monitoring machine components for wear.
- New cycle 271 OCM CONTOUR DATA for defining machining information for the OCM cycles.
- New cycle 272 OCM ROUGHING for machining open pockets while maintaining the tool angle.
- New cycle 273 OCM FINISHING FLOOR for machining open pockets while maintaining the tool angle.
- New cycle 274 OCM FINISHING SIDE for machining open pockets while maintaining the tool angle.
- New cycles 1000 DEFINE RECIP. STROKE, 1001 START RECIP. STROKE, and 1002 STOP RECIP. STROKE for grinding with a reciprocation movement.
- New cycles 1010 DRESSING DIAMETER and 1015 PROFILABRICHTEN for dressing a grinding tool.

- New cycle 1030 ACTIVATE WHEEL EDGE for activating the wheel edges.
- New cycles 1032 GRINDING WHL LENGTH COMPENSATION and 1033 GRINDING WHL RADIUS COMPENSATION for compensating the length and radius of a grinding tool.
- New DATUM TABLE soft key in the **Program Run Single Block** and **Program Run Full Sequence** operating modes.
- In cycles 205 UNIVERSAL PECKING and 241 SINGLE-LIP D.H.DRLNG, the input value for Q379 STARTING POINT is checked and compared to Q201 DEPTH.
- Using cycle 225 ENGRAVING, it is possible to engrave the path or name of an NC program.
- If a limit has been programmed in cycle 233, the FACE MILLING cycle will extend the contour in the infeed direction by the corner radius.
- Cycle 239 ASCERTAIN THE LOAD is only displayed if this has been defined by the machine tool builder.
- The help graphics for Q224 ANGLE OF ROTATION in cycle 256 RECTANGULAR STUD was changed.
- The help graphics for Q326 SPACING IN 1ST AXIS and Q327 SPACING IN 2ND AXIS in cycle 415 PRESET INSIDE CORNER was changed.
- Cycle 444 PROBING IN 3-D logs the measured 3-D distance. Thus, the control can distinguish between scrap and rework.
- The help graphics for Q341 PROBING THE TEETH in cycles 481 and 31 CAL. TOOL LENGTH and in cycles 482 and 32 CAL. TOOL RADIUS was changed.
- In cycles 14xx, it is possible to use a handwheel for pre-positioning in semi-automatic mode. After probing, you can move to clearance height manually.

2

First steps

2.1 Overview

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

The following topics are covered in this chapter:

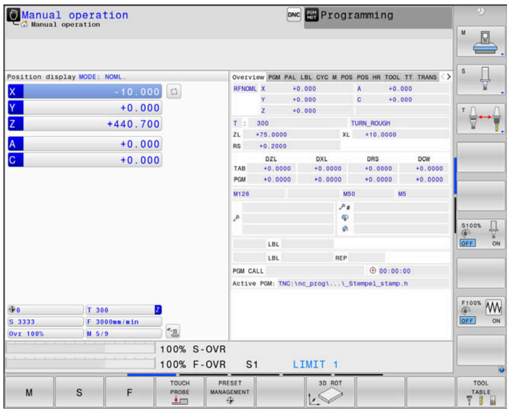
- Switching on the machine
- Programming the workpiece

The following topics are covered in the User's Manual for Setup, Testing and Running NC Programs:

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

2.2 Switching on the machine

Acknowledging the power interruption



⚠ DANGER

Caution: Danger for the operator!

Machines and machine components always present mechanical hazards. Electric, magnetic or electromagnetic fields are particularly hazardous for persons with cardiac pacemakers or implants. The hazard starts when the machine is powered up!

- ▶ Read and follow the machine manual
- ▶ Read and follow the safety precautions and safety symbols
- ▶ Use the safety devices

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

To switch on the machine, proceed as follows:

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

CE

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

I

- ▶ Switch on the machine control voltage
- > The control is in the **Manual operation** mode.

Depending on your machine, you may need to carry out further steps in order to run NC programs.

Further information on this topic

- Switching on the machine
  Further information: User's Manual for Setup, Testing and Running NC Programs

2.3 Programming the first part

Selecting the operating mode

You can write NC programs only in the **Programming** operating mode:

- Press the operating mode key
- > The control switches to the **Programming** operating mode.

Further information on this topic

- Operating modes
  Further information: "Programming", Page 72

Important controls and displays

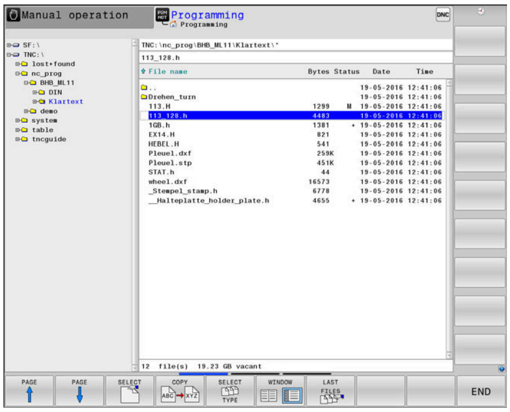
Key	Functions for conversational guidance
ENT	Confirm entry and activate the next dialog prompt
NO ENT	Ignore the dialog question
END	End the dialog immediately
DEL	Abort dialog, discard entries
	Soft keys on the screen with which you select functions appropriate to the active operating state

Further information on this topic

- Writing and editing NC programs
  Further information: "Editing an NC program", Page 97
- Overview of keys
  Further information: "Controls and displays", Page 2

Creating a new NC program / file management

To create a new NC program, proceed as follows:



PGM MGT

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

The file management of the control is arranged much like the file management on a PC with Windows Explorer. The file management enables you to manage data in the control's internal memory.

- ▶ Select a folder
- ▶ Enter the desired file name with the extension **.H**

ENT

- ▶ Press the **ENT** key
- > The control asks for the unit of measure of the new NC program.

MM

- ▶ Press the soft key of the desired unit of measure: **MM** or **INCH**

The control automatically generates the first and last NC blocks of the NC program. You will not be able to change these NC blocks at a later time.

Further information on this topic

- File management
  Further information: "File management", Page 103
- Creating a new NC program
  Further information: "Opening and entering NC programs", Page 89

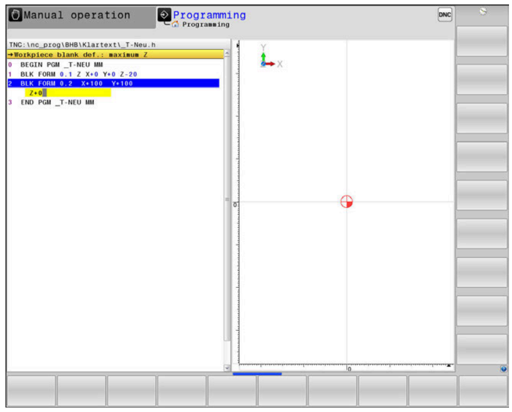
Defining a workpiece blank

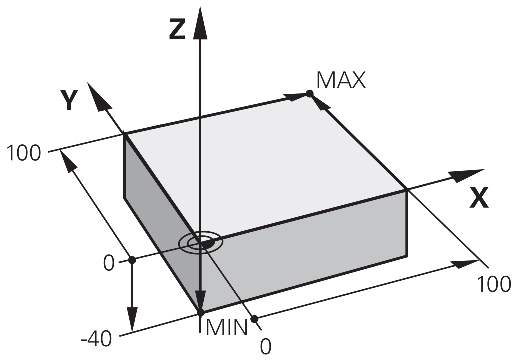
Once you have opened a new NC program, you can define a workpiece blank. You can define a cuboid by entering the MIN and MAX points relative to the selected preset.

After you have selected the desired shape for the blank with the appropriate soft key, the control automatically initiates the workpiece blank definition process and prompts you to enter the required data.

To define a cuboid-shaped blank, proceed as follows:

- ▶ Press the soft key for the desired shape of the workpiece blank
- ▶ **Working plane in graphic: XY?**: Enter the active spindle axis. Z is saved as default setting. Accept with the **ENT** key
- ▶ **Workpiece blank definition: Minimum X**: Enter the smallest X coordinate of the blank relative to the preset (e.g., 0), and confirm with the **ENT** key
- ▶ **Workpiece blank definition: Minimum Y**: Enter the smallest Y coordinate of the blank relative to the preset (e.g., 0), and confirm with the **ENT** key
- ▶ **Workpiece blank definition: Minimum Z**: Enter the smallest Z coordinate of the blank relative to the preset (e.g., –40), and confirm with the **ENT** key
- ▶ **Workpiece blank definition: Maximum X**: Enter the largest X coordinate of the blank relative to the preset (e.g., 100), and confirm with the **ENT** key
- ▶ **Workpiece blank definition: Maximum Y**: Enter the largest Y coordinate of the blank relative to the preset (e.g., 100), and confirm with the **ENT** key
- ▶ **Workpiece blank definition: Maximum Z**: Enter the largest Z coordinate of the blank relative to the preset (e.g., 0), and confirm with the **ENT** key
- > The control ends the dialog.





Example

0 BEGIN PGM NEW MM
1 BLK FORM 0.1 Z X+0 Y+0 Z-40
2 BLK FORM 0.2 X+100 Y+100 Z+0
3 END PGM NEW MM

Further information on this topic

- Define workpiece blank
  Further information: "Creating a new NC program", Page 92

Program layout

NC programs should be arranged consistently in a similar manner. This makes it easier to find your place, accelerates programming and reduces errors.

Recommended program layout for simple, conventional contour machining

Example

```
0 BEGIN PGM  BSPCONT MM
1 BLK FORM 0.1  Z X...  Y... Z...
2 BLK FORM 0.2  X... Y...  Z...
3 TOOL CALL  5 Z  S5000
4 L  Z+250 R0  FMAX M3
5 L  X... Y...  R0 FMAX
6 L  Z+10 R0  F3000 M8
7 APPR ... X... Y...RL F500
...
16 DEP  ... X...  Y... F3000  M9
17 L  Z+250 R0  FMAX M2
18 END PGM  BSPCONT MM
```

1 Call tool, define tool axis
2 Retract the tool; turn on spindle
3 Pre-position the tool in the working plane near the contour starting point
4 Pre-position the tool along the tool axis above the workpiece, or pre-position the tool directly to the cutting depth, and turn on coolant as needed
5 Contour approach
6 Contour machining
7 Contour departure
8 Retract the tool, end the NC program

Further information on this topic

- Contour programming
  Further information: "Programming tool movements for machining", Page 140

Recommended program layout for simple cycle programs

Example

```
0 BEGIN PGM  BSBCYC MM
1 BLK FORM 0.1  Z X...  Y... Z...
2 BLK FORM 0.2  X... Y...  Z...
3 TOOL CALL  5 Z  S5000
4 L  Z+250 R0  FMAX M3
5 PATTERN DEF  POS1( X...  Y... Z...  ) ...
6 CYCL DEF...
7 CYCL CALL PAT  FMAX M8
8 L  Z+250 R0  FMAX M2
9 END PGM  BSBCYC MM
```

1 Call tool, define tool axis
2 Retract the tool; turn on spindle
3 Define the machining positions
4 Define the fixed cycle
5 Call the cycle, and switch on the coolant
6 Retract the tool, end the NC program

Further information on this topic

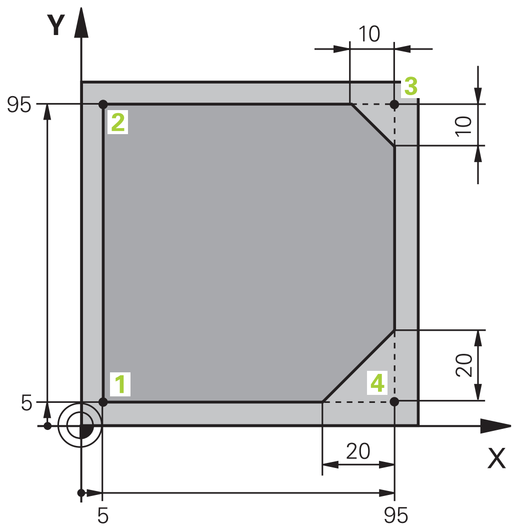
- Cycle programming
  Further information: Cycle Programming User's Manual

Programming a simple contour

Suppose you want to mill a single time around the contour shown on the right at a depth of 5 mm. You have already defined the workpiece blank.

After you have opened an NC block with a function key, the control will prompt you to enter all of the data in the header using dialog texts.

To program the contour, proceed as follows:



Call the tool

TOOL CALL
- Press the **TOOL CALL** key
- Enter the tool data, e.g., tool number 16

ENT
- Press the **ENT** key

ENT
- Confirm the tool axis **Z** with the **ENT** key
- Enter the spindle speed (e.g., 6500)

END
- Press the **END** key
- > The control completes the NC block.

Retract the tool

L
- Press the **L** key

Z
- Press the **Z** axis key
- Enter the retraction value (e.g., 250 mm)

ENT
- Press the **ENT** key

ENT
- At radius compensation, press **ENT**
- > The control applies **R0**, which means there is no radius compensation.

ENT
- At feed rate **F**, press the **ENT** key
- > The control applies **FMAX**.
- If needed, enter a miscellaneous function **M**, such as **M3**, turn on spindle

END
- Press the **END** key
- > The control saves the positioning block.

Pre-position the tool in the working plane

L

- Press the **L** key

X

- Press the **X** axis key
- Enter the value for the position to be approached (e.g., –20 mm)

Y

- Press the **Y** axis key
- Enter the value for the position to be approached (e.g., –20 mm)

ENT

- Press the **ENT** key

ENT

- At radius compensation, press **ENT**
- > The control applies **R0**.

ENT

- At feed rate **F**, press the **ENT** key
- > The control applies **FMAX**.
- If needed, enter a miscellaneous function **M**

END

- Press the **END** key
- > The control saves the positioning block.

Position the tool to the cutting depth

L

- Press the **L** key

Z

- Press the **Z** axis key
- Enter the value for the position to be approached (e.g., -5 mm)

ENT

- Press the **ENT** key

ENT

- At radius compensation, press **ENT**
- > The control applies **R0**.
- Enter the value for the positioning feed rate (e.g., 3000 mm/min)

ENT

- Press the **ENT** key
- Enter a miscellaneous function **M**, such as **M8** to turn coolant on

END

- Press the **END** key
- > The control saves the positioning block.

Approach the contour smoothly

APPR DEP

- Press the **APPR DEP** key
- > The control displays a soft key row with approach and departure functions.

APPR CT

- Press the **APPR CT** soft key
- Enter the coordinates of the contour starting point **1**

ENT

- Press the **ENT** key
- For the center angle **CCA**, enter the approach angle (e.g., 90°)

ENT

- Press the **ENT** key
- Enter the approach radius (e.g., 8 mm)

ENT

- Press the **ENT** key

RL

- Press the **RL** soft key
- > The control applies radius compensation to the left.
- Enter the value for the machining feed rate (e.g., 700 mm/min)

END

- Press the **END** key
- > The control saves the approach movement.

Machine the contour

L

- Press the **L** key
- Enter the changing coordinates of contour point **2** (e.g., **Y 95**)

END

- Press the **END** key
- > The control applies the changed value and retains all of the other information from the previous NC block.

L

- Press the **L** key
- Enter the changing coordinates of contour point **3** (e.g., **X 95**)

END

- Press the **END** key

CHF

- Press the **CHF** key
- Enter the chamfer width (10 mm)

END

- Press the **END** key
- > The control saves the chamfer at the end of the linear block.

L

- Press the **L** key
- Enter the changing coordinates of contour point **4**

END

- Press the **END** key

CHF

- Press the **CHF** key
- Enter the chamfer width (20 mm)

END

- Press the **END** key

Complete the contour with a smooth departure

- Press the **L** key
- Enter the changing coordinates of contour point **1**
- Press the **END** key
- Press the **APPR DEP** key
- Press the **DEP CT** soft key
- For the center angle **CCA**, enter the departure angle (e.g., 90°)
- Press the **ENT** key
- Enter the departure radius (e.g., 8 mm)
- Press the **ENT** key
- Enter the value for the positioning feed rate (e.g., 3000 mm/min)
- Press the **ENT** key
- If needed, enter a miscellaneous function **M**, such as M9, turn off coolant
- Press the **END** key
- > The control saves the departure movement.

Retract the tool

- Press the **L** key
- Press the **Z** axis key
- Enter the retraction value (e.g., 250 mm)
- Press the **ENT** key
- At radius compensation, press **ENT**
- > The control applies **R0**.
- At feed rate **F**, press the **ENT** key
- > The control applies **FMAX**.
- Enter a miscellaneous function **M**, such as **M30** for program end
- Press the **END** key
- > The control saves the positioning block and ends the NC program.

Further information on this topic

- **Complete example with NC blocks**
  Further information: "Example: Linear movements and chamfers with Cartesian coordinates", Page 163
- Creating a new NC program
  Further information: "Opening and entering NC programs", Page 89
- Approaching/departing contours
  Further information: "Approaching and departing a contour", Page 144
- Programming contours
  Further information: "Overview of path functions", Page 154
- Programmable feed rates
  Further information: "Possible feed rate input", Page 95
- Tool radius compensation
  Further information: "Tool radius compensation", Page 134
- Miscellaneous functions M
  Further information: "Miscellaneous functions for program run inspection, spindle and coolant ", Page 225

Creating a cycle program

Suppose that you are tasked with drilling the holes shown to the right with a standard drilling cycle (depth: 20 mm). You have already defined the workpiece blank.

Call the tool

TOOL CALL

- ▶ Press the **TOOL CALL** key
- ▶ Enter the tool data, e.g., tool number 5

ENT

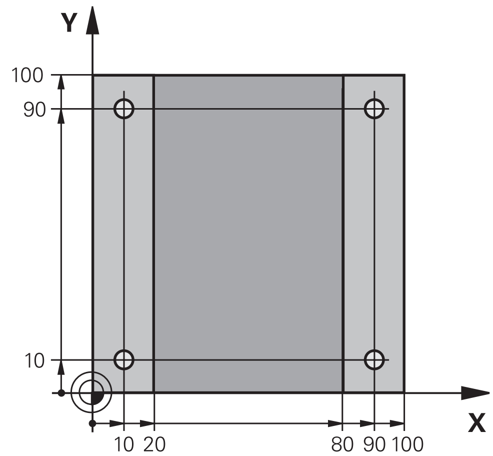
- ▶ Press the **ENT** key

ENT

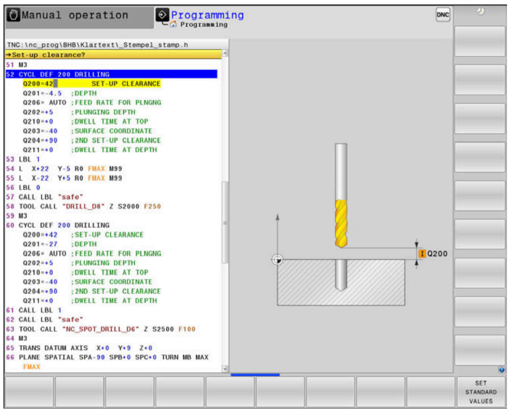
- ▶ Confirm the tool axis **Z** with the **ENT** key
- ▶ Enter the spindle speed (e.g., 4500)

END

- ▶ Press the **END** key
- > The control completes the NC block.



Retract the tool



L

- Press the **L** key

Z

- Press the **Z** axis key
- Enter the retraction value (e.g., 250 mm)

ENT

- Press the **ENT** key

ENT

- At radius compensation, press **ENT**
- > The control applies **R0**, which means there is no radius compensation.

ENT

- At feed rate **F**, press the **ENT** key
- > The control applies **FMAX**.
- If needed, enter a miscellaneous function **M**, such as **M3**, turn on spindle

END

- Press the **END** key
- > The control saves the positioning block.

Define a pattern

SPEC FCT

- Press the **SPEC FCT** key
- > The control opens the soft key row containing the special functions.

CONTOUR + POINT MACHINING

- Press the **CONTOUR + POINT MACHINING** soft key

PATTERN DEF

- Press the **PATTERN DEF** soft key

POINT

- Press the **POINT** soft key
- Enter the coordinates of the first position

ENT

- Confirm each entry with the **ENT** key

ENT

- Press the **ENT** key
- > The control opens the dialog for the next position.
- Enter the coordinates

ENT

- Confirm each entry with the **ENT** key
- Enter the coordinates of all positions

END

- Press the **END** key
- > The control saves the NC block.

Define the cycle

CYCL DEF
- ▶ Press the **CYCL DEF** key

DRILLING/ THREAD
- ▶ Press the **DRILLING/ THREAD** soft key

200
- ▶ Press the **200** soft key
- > The control starts the dialog for cycle definition.
- ▶ Enter the cycle parameters

ENT
- ▶ Confirm each entry with the **ENT** key
- > The control displays a graphic illustrating the respective cycle parameter.

Call the cycle

CYCL CALL
- ▶ Press the **CYCL CALL** key

CYCLE CALL PAT
- ▶ Press the **CYCLE CALL PAT** soft key

ENT
- ▶ Press the **ENT** key
- > The control applies **FMAX**.
- ▶ If needed, enter a miscellaneous function **M**

END
- ▶ Press the **END** key
- > The control saves the NC block.

Retract the tool

L
- ▶ Press the **L** key

Z
- ▶ Press the **Z** axis key
- ▶ Enter the retraction value (e.g., 250 mm)

ENT
- ▶ Press the **ENT** key

ENT
- ▶ At radius compensation, press **ENT**
- > The control applies **R0**.

ENT
- ▶ At feed rate **F**, press the **ENT** key
- > The control applies **FMAX**.
- ▶ Enter a miscellaneous function **M**, such as **M30** for program end

END
- ▶ Press the **END** key
- > The control saves the positioning block and ends the NC program.

Example

0 BEGIN PGM C200 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-40	Workpiece blank definition
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 5 Z S4500	Tool call
4 L Z+250 R0 FMAX M3	Retract the tool; turn on spindle
5 PATTERN DEF POS1 (X+10 Y+10 Z+0) POS2 (X+10 Y+90 Z+0) POS3 (X+90 Y+90 Z+0) POS4 (X+90 Y+10 Z+0)	Define the machining positions
6 CYCL DEF 200 DRILLING	Define the cycle
Q200=2 ;SET-UP CLEARANCE	
Q201=-20 ;DEPTH	
Q206=250 ;FEED RATE FOR PLNGNG	
Q202=5 ;PLUNGING DEPTH	
Q210=0 ;DWELL TIME AT TOP	
Q203=-10 ;SURFACE COORDINATE	
Q204=20 ;2ND SET-UP CLEARANCE	
Q211=0.2 ;DWELL TIME AT DEPTH	
Q395=0 ;DEPTH REFERENCE	
7 CYCL CALL PAT FMAX M8	Turn on coolant; call cycle
8 L Z+250 R0 FMAX M30	Retract the tool, end program
9 END PGM C200 MM	

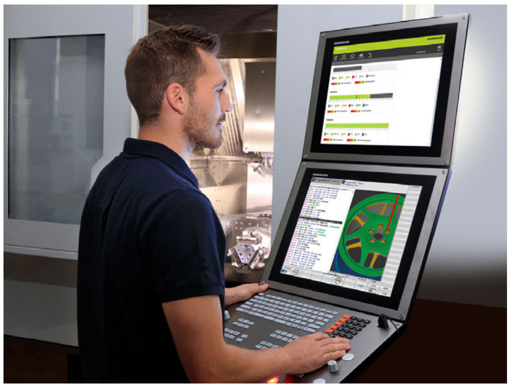
Further information on this topic

- Creating a new NC program
  Further information: "Opening and entering NC programs", Page 89
- Cycle programming
  Further information: Cycle Programming User's Manual

3

Fundamentals

3.1 The TNC 640



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

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

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

HEIDENHAIN Klartext and DIN/ISO

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

It is also possible to program in ISO format.

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

Compatibility

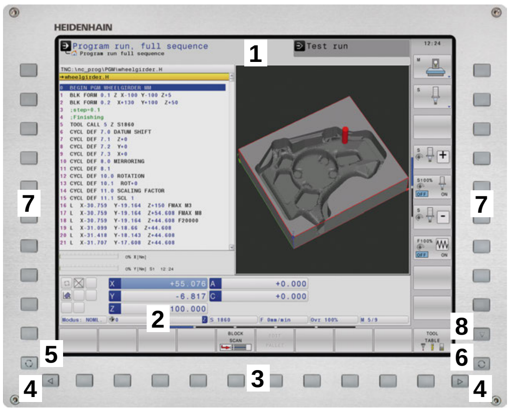
NC programs created on HEIDENHAIN contouring controls (starting from the TNC 150 B) may not always run on the TNC 640. If the NC blocks contain invalid elements, the control will mark these as ERROR blocks or with error messages when the file is opened.

> Please also note the detailed description of the differences between the iTNC 530 and the TNC 640.
> **Further information:** "Differences between the TNC 640 and the iTNC 530", Page 598

3.2 Visual display unit and operating panel

Display screen

The control is shipped with a 19-inch screen.



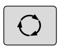
1 Header
   When the control is on, the screen displays the selected operating modes in the header: The machine operating mode at left and the programming mode at right. The currently active mode is displayed in the larger field of the header, where the dialog prompts and messages also appear (exception: if the control only displays graphics).
2 Soft keys
   In the footer the control indicates additional functions in a soft-key row. You can select these functions by pressing the keys immediately below them. The thin bars immediately above the soft-key row indicate the number of soft-key rows that can be called with the keys to the right and left that are used to switch the soft keys. The bar representing the active soft-key row is blue
3 Soft-key selection keys
4 Keys for switching the soft keys
5 Setting the screen layout
6 Key for switchover between machine operating modes, programming modes, and a third desktop
7 Soft-key selection keys for machine tool builders
8 Keys for switching the soft keys for machine tool builders

> If you are using a TNC 640 with touch control, you can replace some keystrokes with gestures.
> **Further information:** "Operating the touchscreen", Page 545

Setting the screen layout

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

Setting the screen layout:



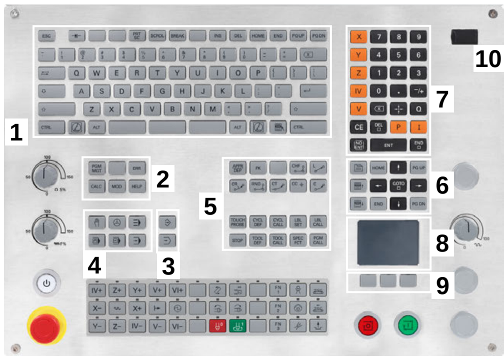
- Press the **screen layout** key: The soft-key row shows the available layout options
  Further information: "Modes of operation", Page 71

PROGRAM + GRAPHICS

- Select the desired screen layout with a soft key

Control panel

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



1 Alphabetic keyboard for entering texts and file names, as well as for ISO programming
2
- File management
- Calculator
- MOD function
- HELP function
- Show error messages
- Toggle between the operating modes

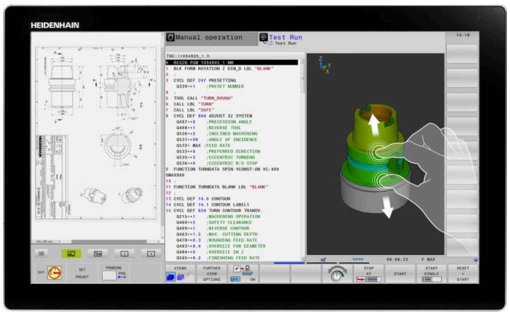
3 Programming modes
4 Machine operating modes
5 Initiating programming dialogs
6 Navigation keys and **GOTO** jump command
7 Numerical input and axis selection
8 Touchpad
9 Mouse buttons
10 USB connection

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

> If you are using a TNC 640 with touch control, you can replace some keystrokes with gestures.
>
> **Further information:** "Operating the touchscreen", Page 545

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

Extended Workspace Compact



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

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

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

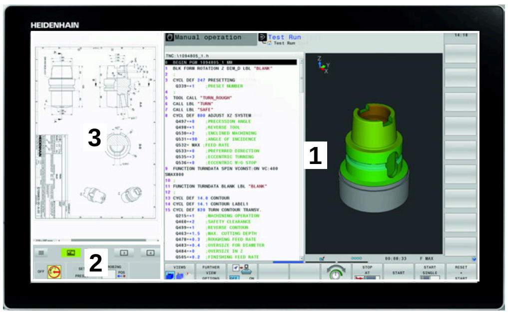
The additional screen workspace in **Extended Workspace Compact** mode provides full multitouch support. When you switch to full-screen mode, you can use the HEIDENHAIN keyboard for your external applications.

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

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

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

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



Extended Workspace Compact is divided into three areas:

1 **JH Standard**:

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

2 **JH Extended**:

This area provides configurable quick accesses to HEIDENHAIN applications.

Contents of **JH Extended:**

- **HEROS** menu
- 1st screen workspace, **Manual Operation** mode
- 2nd screen workspace, **Programming** operating mode
- 3rd and 4th screen workspaces, freely usable for applications, such as the **CAD Converter**
- Collection of frequently used soft keys

> Benefits of **JH Extended:**
> - Each operating mode has its own additional soft-key row
> - Navigation through the various rows of HEIDENHAIN soft keys is no longer necessary

3 **OEM**:

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

Contents of the **OEM** area:

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

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

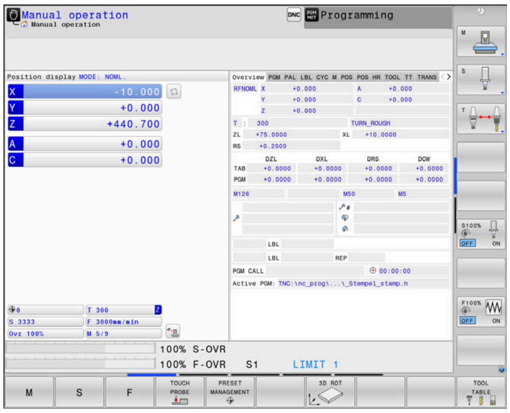
In machine parameter **CfgSideScreen** (no. 130000), you can select the connection to be embedded in the second workspace on the screen.

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

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

3.3 Modes of operation

Manual Operation and El. Handwheel



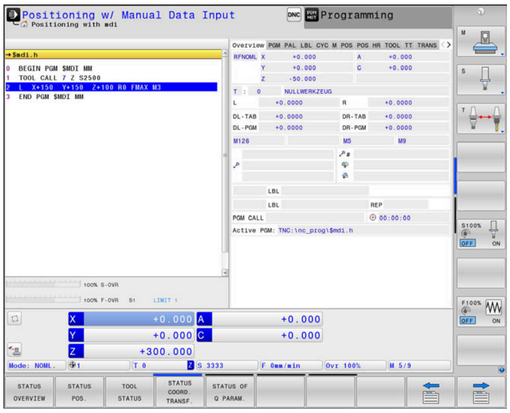
The **Manual operation** mode is required for setting up the machine tool. In this mode of operation, you can position the machine axes manually or by increments, set the presets and tilt the working plane.

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

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

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

Positioning with Manual Data Input



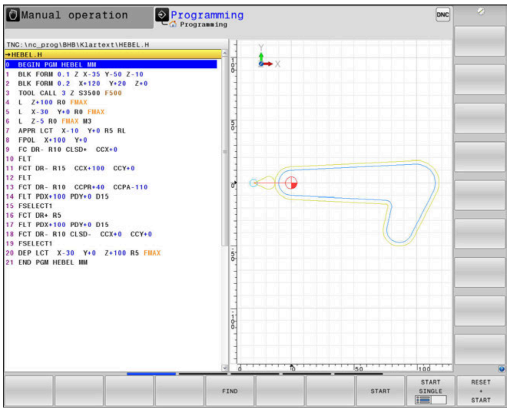
This mode of operation is used for programming simple traversing movements, such as for face milling or pre-positioning.

Soft keys for selecting the screen layout

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

Programming

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

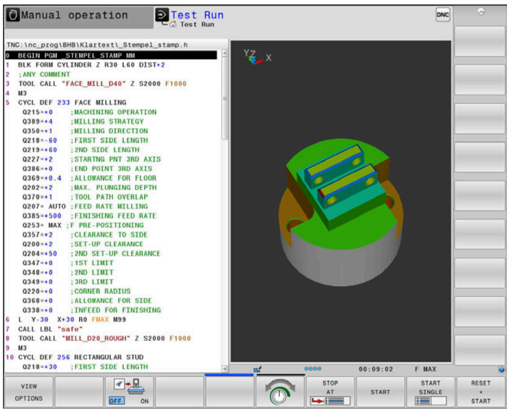


Soft keys for selecting the screen layout

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

Test Run

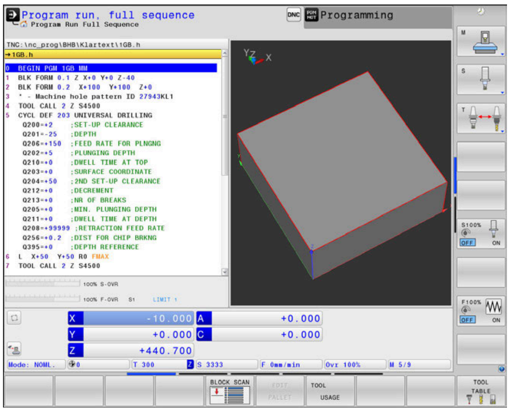
In the **Test Run** operating mode, the control checks NC programs and program sections for errors, such as geometrical incompatibilities, missing or incorrect data within the NC program or violations of the working space. This simulation is supported graphically in different display modes.



Soft keys for selecting the screen layout

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

Program Run, Full Sequence and Program Run, Single Block



In the **Program Run Full Sequence** operating mode, the control runs an NC program continuously to its end or to a manual or programmed stop. You can resume program run after an interruption.

In the **Program Run Single Block** operating mode, you execute each NC block separately by pressing the **NC start** key. With point pattern cycles and **CYCL CALL PAT**, the control stops after each point.

Soft keys for selecting the screen layout

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

Soft keys for screen layout with pallet tables

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

3.4 NC fundamentals

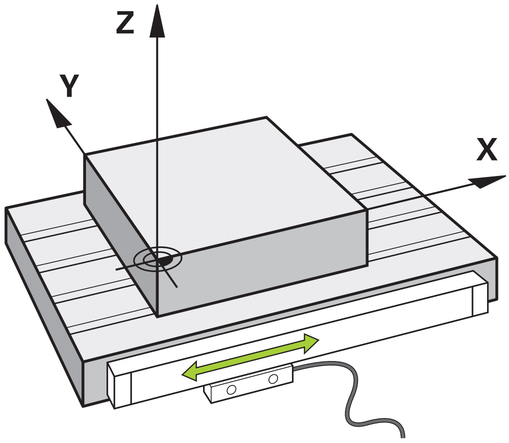
Position encoders and reference marks

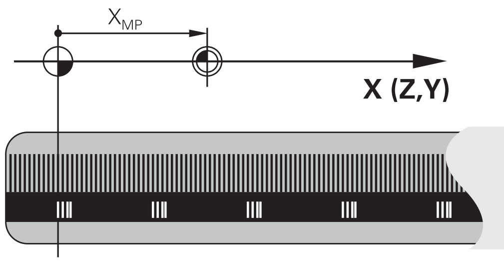
The machine axes are equipped with position encoders that register the positions of the machine table or tool. Linear axes are usually equipped with linear encoders, rotary tables and tilting axes with angle encoders.

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

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

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



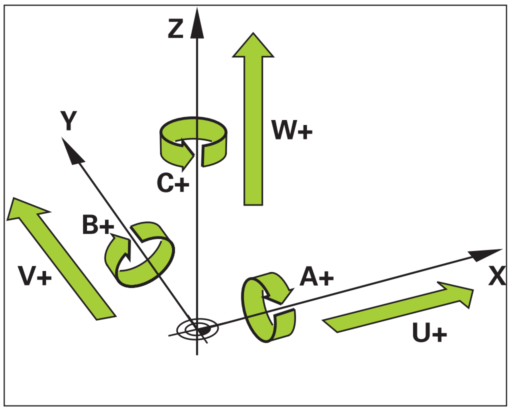


Programmable axes

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

The designations of the programmable axes are given in the table below.

Principal axis	Parallel axis	Rotary axis
X	U	A
Y	V	B
Z	W	C



Refer to your machine manual!

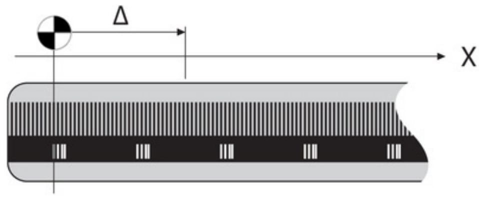
The number, designation and assignment of the programmable axes depend on the machine.

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

Reference systems

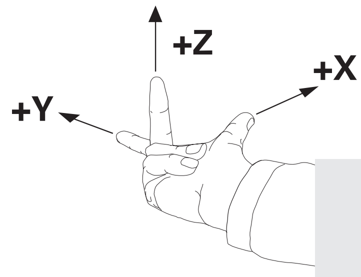
For the control to move an axis in accordance with a defined path, it requires a **reference system**.

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



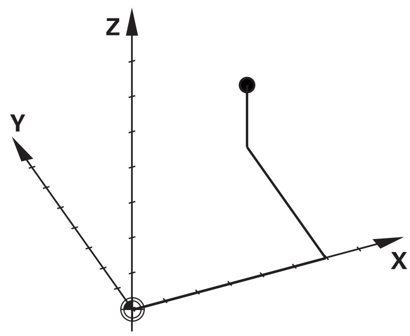
To approach a point on the **plane**, the control requires two axes and therefore a reference system with two dimensions.

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



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

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



In order, for example, for the control to always perform a tool change at the same position, as well as always execute a machining operation referenced to the current workpiece position, the control must be able to differentiate between different reference systems.

The control differentiates between the following reference systems:

- Machine coordinate system M-CS:
  Machine **C**oordinate **S**ystem
- Basic coordinate system B-CS:
  Basic **C**oordinate **S**ystem
- Workpiece coordinate system W-CS:
  Workpiece **C**oordinate **S**ystem
- Working plane coordinate system WPL-CS:
  Working **Pl**ane **C**oordinate **S**ystem
- Input coordinate system I-CS:
  Input **C**oordinate **S**ystem
- Tool coordinate system T-CS:
  Tool **C**oordinate **S**ystem

> All reference systems build upon each other. They are subject to the kinematic chain of the specific machine tool.
>
> The machine coordinate system is the reference system.

Machine coordinate system M-CS

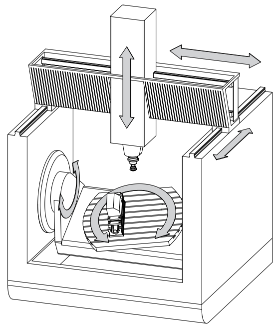
The machine coordinate system corresponds to the description of kinematics and therefore to the actual mechanical design of the machine tool.

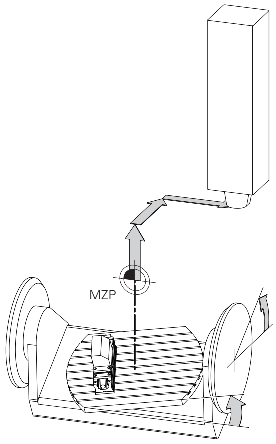
Because the mechanics of a machine tool never precisely correspond to a Cartesian coordinate system, the machine coordinate system consists of several one-dimensional coordinate systems. These one-dimensional coordinate systems correspond to the physical machine axes that are not necessarily perpendicular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

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

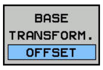
The position of the coordinate origin, the machine datum, is defined by the machine manufacturer during machine configuration. The values in the machine configuration define the zero positions of the encoders and the corresponding machine axes. The machine datum does not necessarily have to be located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

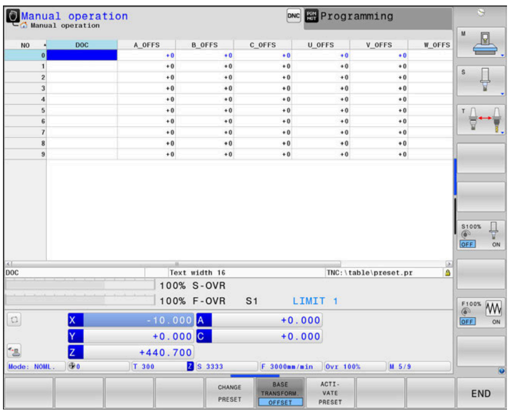
Because the machine configuration values cannot be modified by the user, the machine coordinate system is used for determining constant positions, e.g. the tool change point.





Machine datum (MZP)

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



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

Further information: User's Manual for Setup, Testing and Running NC Programs

NOTICE

Danger of collision!

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

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

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

> Another feature is **OEM-OFFSET**, which is available only to the machine tool builder. **OEM-OFFSET** can be used to define additive axis shifts for rotary and parallel axes.
>
> The sum of all **OFFSET** values (from all the above **OFFSET** input possibilities) results in the difference between the **ACTL.** position and the **RFACTL** position of an axis.

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

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

- In the **Positioning w/ Manual Data Input** operating mode, run an NC block with **L IY+10**
- > The control determines the required axis nominal values from the defined values.
- > During positioning the control moves the **Y and Z** machine axes.
- > The **RFACTL** and **RFNOML** displays show movements of the Y axis and Z axis in the machine coordinate system.
- > The **ACTL.** and **NOML.** displays only show one movement of the Y axis in the input coordinate system.
- In the **Positioning w/ Manual Data Input** operating mode, run an NC block with **L IY-10 M91**
- > The control determines the required axis nominal values from the defined values.
- > During positioning the control only moves the **Y** machine axis.
- > The **RFACTL** and **RFNOML** displays only show one movement of the Y axis in the machine coordinate system.
- > The **ACTL.** and **NOML.** displays show movements of the Y axis and Z axis in the input coordinate system.

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

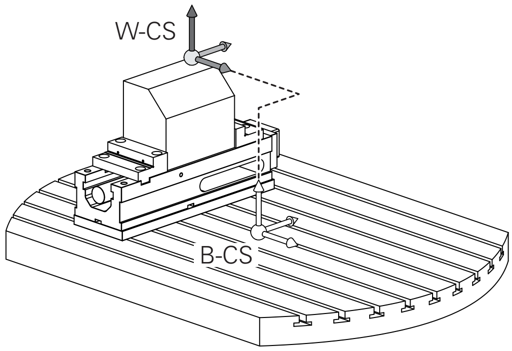
Basic coordinate system B-CS

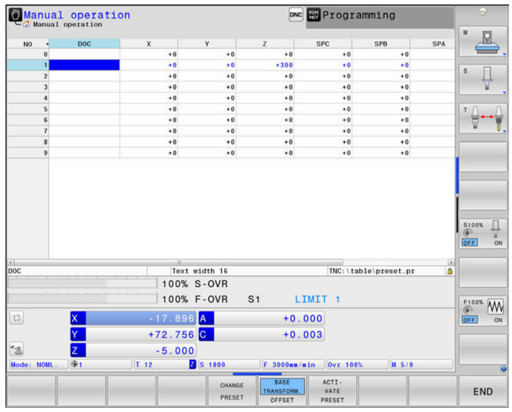
The basic coordinate system is a 3-D Cartesian coordinate system. Its coordinate origin is the end of the kinematics model.

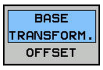
The orientation of the basic coordinate system in most cases corresponds to that of the machine coordinate system. There may be exceptions to this if a machine manufacturer uses additional kinematic transformations.

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

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





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

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

Further information: User's Manual for Setup, Testing and Running NC Programs

NOTICE

Danger of collision!

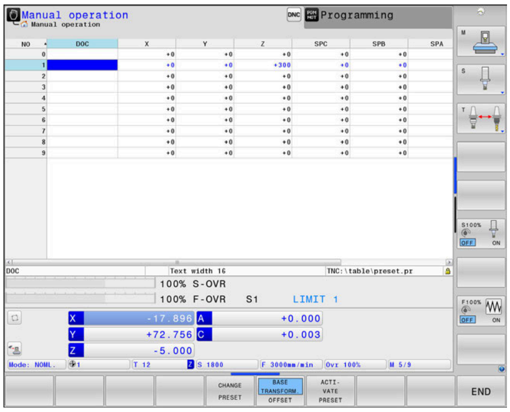
Your control may feature an additional pallet preset table, depending on the machine. In this table the machine tool builder can define **BASE TRANSFORM.** values that take effect before the **BASE TRANSFORM.** values you specify in the preset table become effective. The **PAL** tab of the additional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

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

Workpiece coordinate system W-CS

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

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

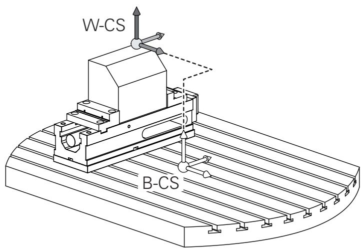


Soft key	Application
BASE TRANSFORM. OFFSET	The user determines the position and orientation of the workpiece coordinate system by using a 3-D touch probe for example. The control saves the values determined with respect to the basic coordinate system as **BASE TRANSFORM.** values in the preset management.

Further information: User's Manual for Setup, Testing and Running NC Programs

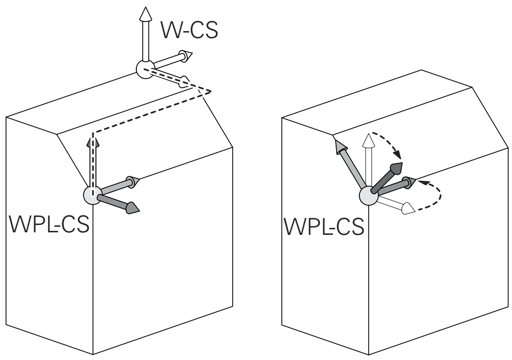
> The **Global Program Settings** function (Option 44) additionally provides the following transformations:
>
> - The **Additive basic rotat. (W-CS)** is added to a basic rotation or a 3-D basic rotation from the preset table and the pallet preset table. The **Additive basic rotat. (W-CS)** is the first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

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



Transformations in the workpiece coordinate system:

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



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

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

Programming notes:

- Transformations (mirroring and shifting) that are programmed before the **PLANE** functions (except for **PLANE AXIAL**) will change the position of the tilt datum (origin of the working plane coordinate system WPL-CS) and the orientation of the rotary axes
  - If you just program a shift, then only the position of the tilt datum will change
  - If you just program mirroring, then only the orientation of the rotary axes will change
- When used in conjunction with **PLANE AXIAL** and Cycle 19, the programmed transformations (mirroring, rotation and scaling) do not affect the position of the tilt datum or the orientation of the rotary axes

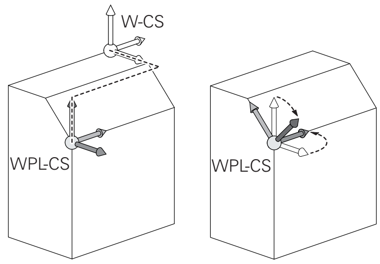
Without active transformations in the workpiece coordinate system, the position and orientation of the working plane coordinate system and workpiece coordinate system are identical.

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

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

Further information: "Working plane coordinate system WPL-CS", Page 81

Working plane coordinate system WPL-CS

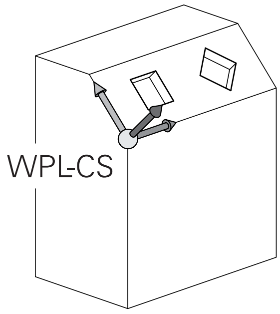


The working plane coordinate system is a 3-D Cartesian coordinate system.

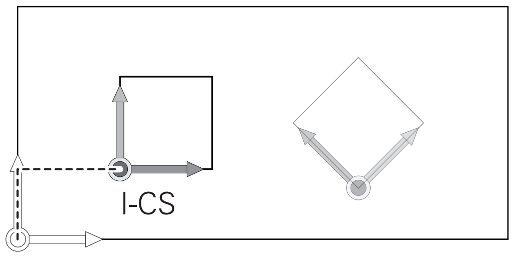
The position and orientation of the working plane coordinate system depend on the active transformations in the workpiece coordinate system.

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

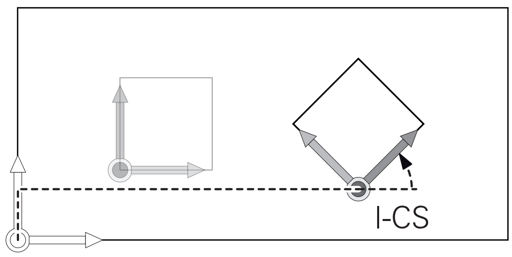
In the working plane coordinate system the user defines the position and orientation of the input coordinate system with use of transformations.



> The **Mill-Turning** function (option 50) additionally provides the **OEM rotation** and **precession angle** transformations.
>
> - **OEM rotation** is available only to the machine tool builder and takes effect before the **precession angle**
> - **Precession angle** is defined in Cycles 800 **ADJUST XZ SYSTEM**, 801 **RESET ROTARY COORDINATE SYSTEM** and 880 **GEAR HOBBING**, and takes effect before the other transformations of the working plane coordinate system
>
> The active values of the two transformations (if not equal to 0) are shown on the **POS** tab of the additional status display. Check the values also in milling mode because any active transformations will also remain active in that mode!



> Refer to your machine manual!
>
> Your machine tool builder can use the **OEM rotation** and **precession angle** transformations also without the **Mill-Turning** function (option 50).



Transformations in the working plane coordinate system:

- Cycle 7 **DATUM SHIFT**
- Cycle 8 **MIRROR IMAGE**
- Cycle 10 **ROTATION**
- Cycle 11 **SCALING**
- Cycle 26 **AXIS-SPECIFIC SCALING**
- **PLANE RELATIVE**

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

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

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

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

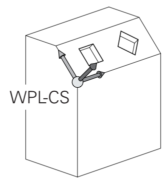
Without active transformations in the working plane coordinate system, the position and orientation of the input coordinate system and working plane coordinate system are identical.

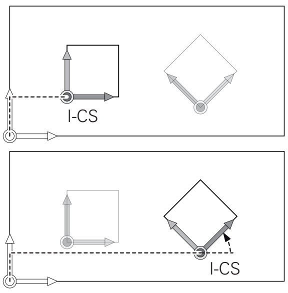
In addition, there are no transformations in the workpiece coordinate system on 3-axis machine tools or with pure 3-axis machining. The **BASE TRANSFORM.** values of the active line of the preset table have a direct effect on the input coordinate system with this assumption.

Input coordinate system I-CS

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

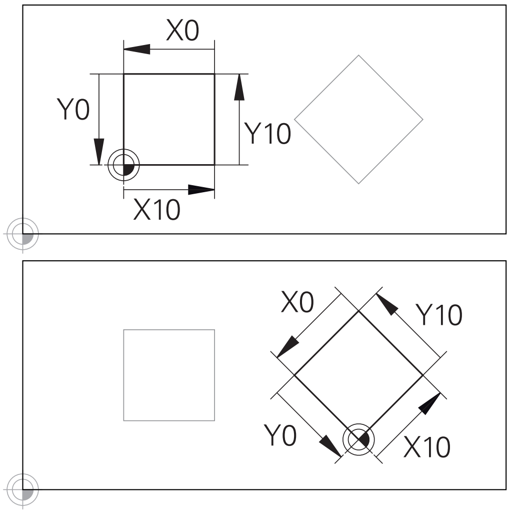
The position and orientation of the input coordinate system depend on the active transformations in the working plane coordinate system.





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

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



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

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

Positioning blocks in input coordinate system:

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

Example

```
7 X+48 R+
```

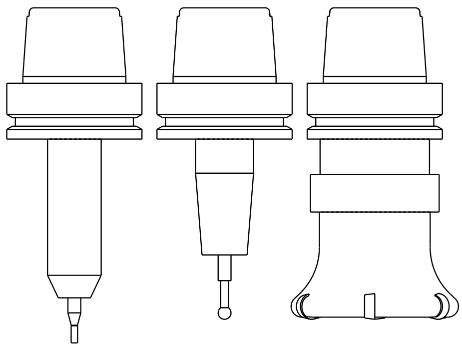
```
7 L X+48 Y+102 Z-1.5 R0
```

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

> The position of the tool coordinate system is determined by the Cartesian coordinates X, Y and Z also for positioning blocks with surface normal vectors.
>
> In conjunction with 3-D tool compensation, the position of the tool coordinate system can be shifted along the surface normal vectors.

> Orientation of the tool coordinate system can be performed in various reference systems.
>
> **Further information:** "Tool coordinate system T-CS", Page 84

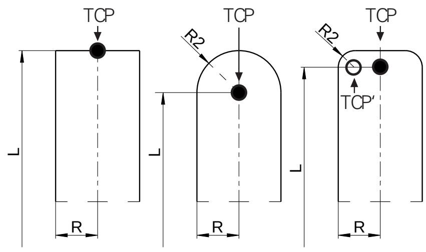
Tool coordinate system T-CS



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

Further information: User's Manual for Setup, Testing and Running NC Programs

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

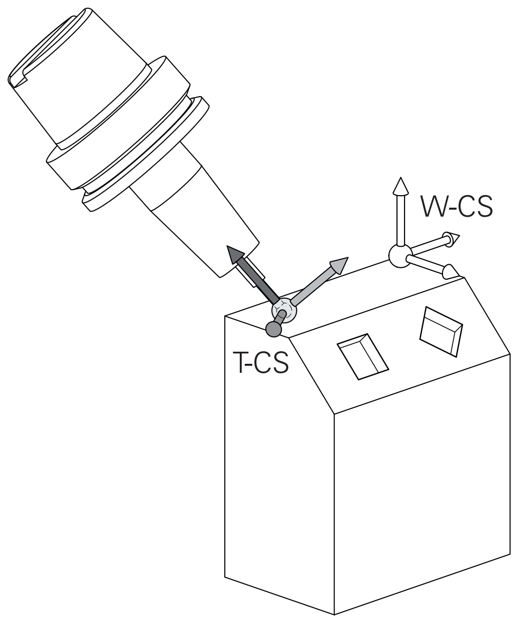


In accordance with the values from the tool table, the coordinate origin of the tool coordinate system is shifted to the tool center point TCP. TCP stands for **T**ool **C**enter **P**oint.

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

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

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



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

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

Tool angle of inclination in the machine coordinate system:

Example

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

Tool angle of inclination in the working plane coordinate system:

Example

```
6 FUNCTION TCPM F TCP AXIS SPAT PATHCTRL AXIS
7 L A+0 B+45 C+0 R0 F2500
```

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

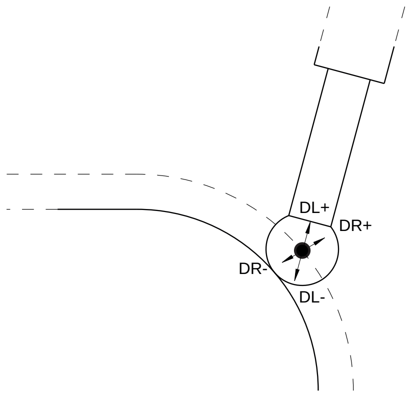
```
7 LN X+48 Y+102 Z-1.5 NX-0.04658107 NY0.00045007
  NZ0.8848844 R0 M128
```

With the shown positioning blocks with vectors, 3-D tool compensation is possible with compensation values **DL**, **DR** and **DR2** from the **TOOL CALL** block or from the **.tco** compensation table.

The methods of function of the compensation values depend on the type of tool.

The control detects the various tool types with the columns **L**, **R** and **R2** of the tool table:

- $R2_{TAB} + DR2_{TAB} + DR2_{PROG} = 0$
  → end mill
- $R2_{TAB} + DR2_{TAB} + DR2_{PROG} = R_{TAB} + DR_{TAB} + DR_{PROG}$
  → radius cutter or ball cutter
- $0 < R2_{TAB} + DR2_{TAB} + DR2_{PROG} < R_{TAB} + DR_{TAB} + DR_{PROG}$
  → toroid cutter or toroidal cutter



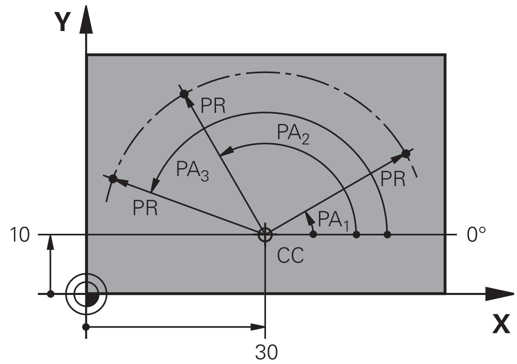
Without the **TCPM** function or miscellaneous function **M128**, orientation of the tool coordinate system and input coordinate system is identical.

Designation of the axes on milling machines

The X, Y and Z axes on your milling machine are also referred to as tool axis, principal axis (1st axis) and secondary axis (2nd axis). The assignment of the tool axis is decisive for the assignment of the principal and secondary axes.

Tool axis	Principal axis	Secondary axis
X	Y	Z
Y	Z	X
Z	X	Y

Polar coordinates

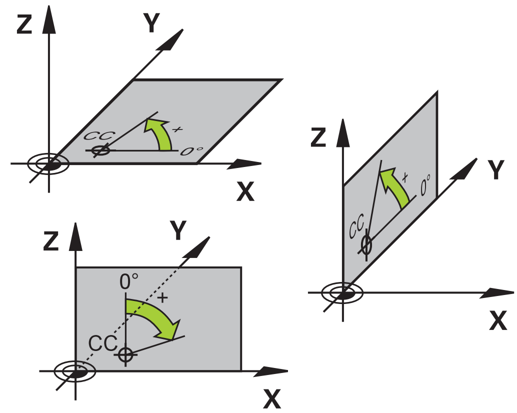


If the production drawing is dimensioned in Cartesian coordinates, you write the NC program using Cartesian coordinates. For parts containing circular arcs or angles, it is often simpler to give the dimensions in polar coordinates.

While the Cartesian coordinates X, Y and Z are three-dimensional and can describe points in space, polar coordinates are two-dimensional and describe points in a plane. Polar coordinates have their datum at a circle center (CC), or pole. A position in a plane can be clearly defined by the:

- Polar Radius, the distance from the circle center CC to the position, and the
- Polar Angle, the value of the angle between the angle reference axis and the line that connects the circle center CC with the position.

Setting the pole and the angle reference axis



The pole is set by entering two Cartesian coordinates in one of the three planes. These coordinates also set the reference axis for the polar angle PA.

Coordinates of the pole (plane)	Angle reference axis
X/Y	+X
Y/Z	+Y
Z/X	+Z

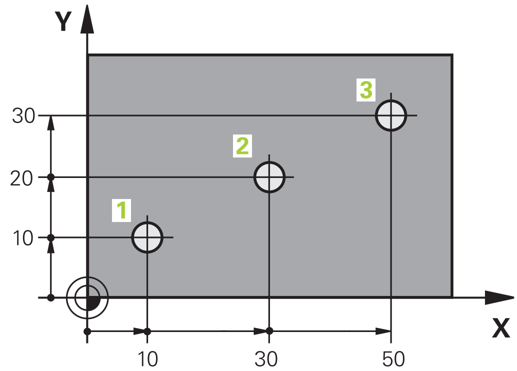
Absolute and incremental workpiece positions

Absolute workpiece positions

Absolute coordinates are position coordinates that are referenced to the datum of the coordinate system (origin). Each position on the workpiece is unambiguously defined by its absolute coordinates.

Example 1: Holes dimensioned in absolute coordinates

Hole **1**	Hole **2**	Hole **3**
X = 10 mm	X = 30 mm	X = 50 mm
Y = 10 mm	Y = 20 mm	Y = 30 mm

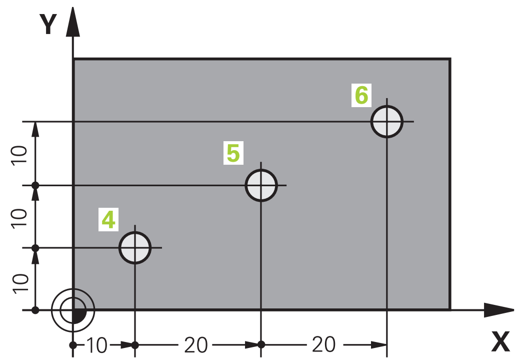


Incremental workpiece positions

Incremental coordinates are referenced to the last programmed nominal position of the tool, which serves as the relative (imaginary) datum. When you write an NC program in incremental coordinates, you thus program the tool to move by the distance between the previous and the subsequent nominal positions. This is why they are also referred to as chain dimensions.

To program a position in incremental coordinates, enter the letter **I** before the axis.

Example 2: Holes dimensioned in incremental coordinates



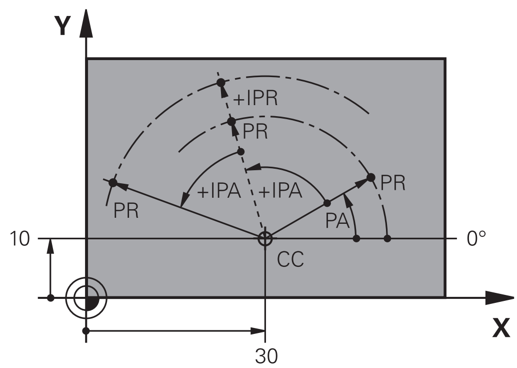
Absolute coordinates of hole **4**
X = 10 mm
Y = 10 mm

Hole **5**, with respect to **4**	Hole **6**, with respect to **5**
X = 20 mm	X = 20 mm
Y = 10 mm	Y = 10 mm

Absolute and incremental polar coordinates

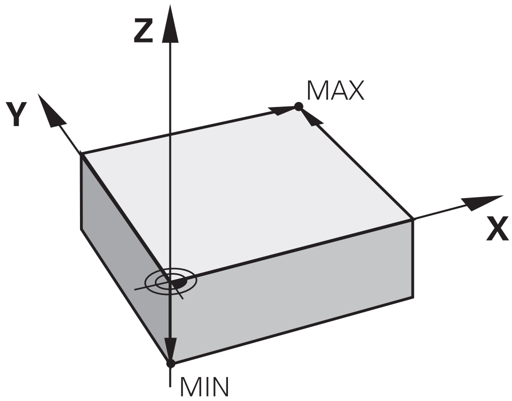
Absolute coordinates always refer to the pole and the angle reference axis.

Incremental polar coordinates always refer to the last programmed nominal position of the tool.



Selecting the preset

A production drawing specifies a certain form element of the workpiece (usually a corner) as the absolute reference point (datum). When setting the preset, first align the workpiece along the machine axes, and move the tool to a known position in each axis relative to the workpiece. For each position, set the display of the control either to zero or to a known position value. You thereby assign the workpiece to the reference system that is applicable for the control's display or your NC program.



If the production drawing is dimensioned in relative reference points, simply use the coordinate transformation cycles.

Further information: Cycle Programming User's Manual

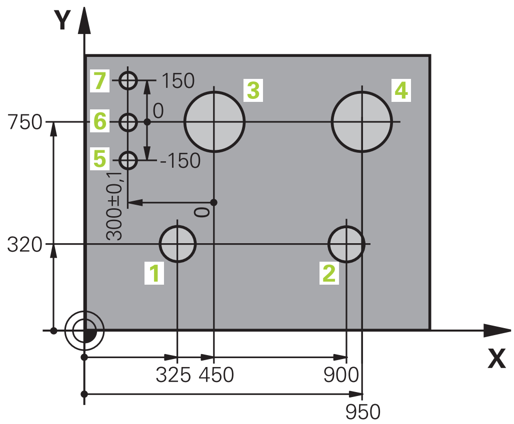
If the production drawing is not dimensioned for NC programming, then select a position or corner of the workpiece as a reference point from which the dimensions of the remaining workpiece positions can be determined.

A particularly convenient way of setting the presets is with a 3-D touch probe from HEIDENHAIN.

Further information: User's Manual for Setup, Testing and Running NC Programs

Example

The workpiece drawing shows holes (**1** to **4**), whose dimensions are shown relative to an absolute reference point with the coordinates X=0 Y=0. The coordinates of holes **5** to **7** are measured relative to a reference point with the absolute coordinates X=450 Y=750. The **Datum shift** cycle allows you to temporarily shift the datum to the position X=450, Y=750 in order for you to program the holes (**5** to **7**) without further calculations.



3.5 Opening and entering NC programs

Structure of an NC program in HEIDENHAIN Klartext

An NC program consists of a series of NC blocks. The illustration at right shows the elements of an NC block.

The control numbers the NC blocks of an NC program in ascending sequence.

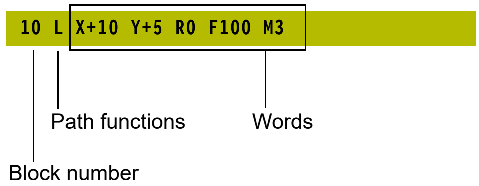
The first NC block of an NC program is identified by **BEGIN PGM**, the program name, and the active unit of measure.

The subsequent NC blocks contain information on

- The workpiece blank
- Tool calls
- Approaching a safe position
- Feed rates and spindle speeds, as well as
- Path contours, cycles and other functions

The last block of a program is identified by **END PGM**, the program name, and the active unit of measure.

NC block



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 after a tool change!

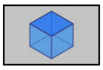
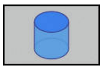
- If necessary, program an additional safe auxiliary position

Defining the blank: BLK FORM

Immediately after opening a new NC program, you define an unmachined workpiece blank. If you wish to define the blank at a later stage, then press the **SPEC FCT** key, the **PROGRAM DEFAULTS** soft key, and then the **BLK FORM** soft key. The control needs this definition for its graphical simulations.

> You only need to define the workpiece blank if you wish to run a graphic test for the NC program.

The control can depict various types of blank forms:

Soft key	Function
	Define a rectangular blank
	Define a cylindrical blank
	Define a rotationally symmetric blank of any shape

Rectangular blank

The sides of the cuboid lie parallel to the X, Y and Z axes. This blank is defined by two of its corner points:

- MIN point: the smallest X, Y and Z coordinates of the blank form, entered as absolute values.
- MAX point: the largest X, Y and Z coordinates of the blank form, entered as absolute or incremental values

Example

0 BEGIN PGM NEW MM	Program begin, name, unit of measure
1 BLK FORM 0.1 Z X+0 Y+0 Z-40	Spindle axis, MIN point coordinates
2 BLK FORM 0.2 X+100 Y+100 Z+0	MAX point coordinates
3 END PGM NEW MM	Program end, name, unit of measure

Cylindrical blank

The cylindrical blank form is defined by the dimensions of the cylinder:

- X, Y or Z: Rotation axis
- D, R: Diameter or radius of the cylinder (with positive algebraic sign)
- L: Length of the cylinder (with positive algebraic sign)
- DIST: Shifting along the rotational axis
- DI, RI: Inside diameter or inside radius for a hollow cylinder

> The parameters **DIST** and **RI** or **DI** are optional and need not be programmed.

Example

0 BEGIN PGM NEW MM	Program begin, name, unit of measure
1 BLK FORM CYLINDER Z R50 L105 DIST+5 RI10	Spindle axis, radius, length, distance, inside radius
2 END PGM NEW MM	Program end, name, unit of measure

Rotationally symmetric blank of any shape

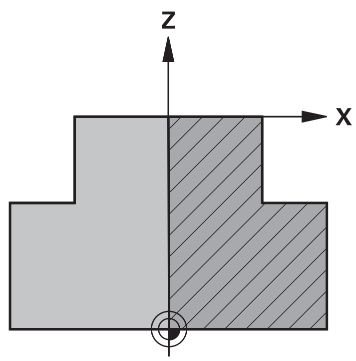
You define the contour of the rotationally symmetric blank in a subprogram. Use X, Y or Z as the rotation axis.

In the workpiece blank definition you refer to the contour description:

- DIM_D, DIM-R: Diameter or radius of the rotationally symmetrical blank form
- LBL: Subprogram with the contour description

The contour description may contain negative values in the rotation axis but only positive values in the reference axis. The contour must be closed, i.e. the contour beginning corresponds to the contour end.

If you define a rotationally symmetric blank with incremental coordinates, the dimensions are then independent of the diameter programming.



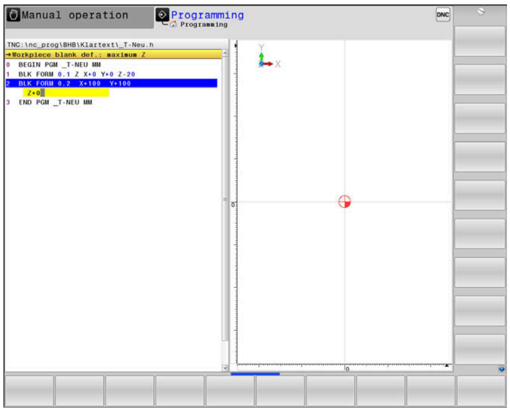
> The subprogram can be designated with a number, an alphanumeric name, or a QS parameter.

Example

0 BEGIN PGM NEW MM	Program begin, name, unit of measure
1 BLK FORM ROTATION Z DIM_R LBL1	Spindle axis, manner of interpretation, subprogram number
2 M30	End of main program
3 LBL 1	Subprogram start
4 L X+0 Z+1	Starting point of contour
5 L X+50	Programming in the positive direction of the principal axis
6 L Z-20	
7 L X+70	
8 L Z-100	
9 L X+0	
10 L Z+1	Contour end
11 LBL 0	End of subprogram
12 END PGM NEW MM	Program end, name, unit of measure

Creating a new NC program

An NC program is always entered in **Programming** mode. Example for creating a program:



- Operating mode: Press the **Programming** key

PGM MGT

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

Select the directory in which you wish to store the new NC program:

FILE NAME = NEW.H

ENT

- Enter the new program name
- Press the **ENT** key

MM

- Select the unit of measure: Press the **MM** or **INCH** soft key
- > The control switches the screen layout and initiates the dialog for defining the **BLK FORM** (workpiece blank).
- Select a rectangular workpiece blank: Press the soft key for a rectangular blank form

Working plane in graphic: XY

Z

- Enter the spindle axis, e.g. **Z**

Workpiece blank def.: Minimum

ENT

- Enter in sequence the X, Y and Z coordinates of the MIN point and confirm each of your entries with the **ENT** key

Workpiece blank def.: Maximum

ENT

- Enter in sequence the X, Y and Z coordinates of the MAX point and confirm each of your entries with the **ENT** key

Example

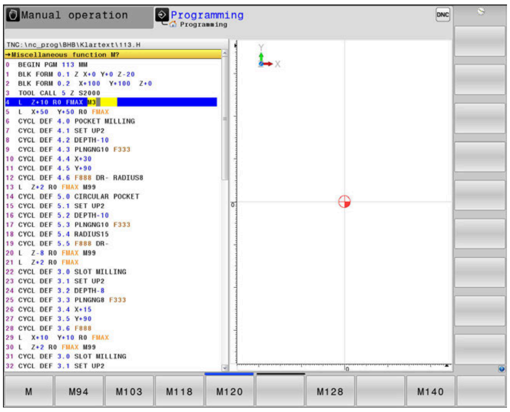
0 BEGIN PGM NEW MM	Program begin, name, unit of measure
1 BLK FORM 0.1 Z X+0 Y+0 Z-40	Spindle axis, MIN point coordinates
2 BLK FORM 0.2 X+100 Y+100 Z+0	MAX point coordinates
3 END PGM NEW MM	Program end, name, unit of measure

The control automatically generates the block numbers as well as the **BEGIN** and **END** blocks.

> If you do not wish to define a workpiece blank, then cancel the dialog at **Working plane in graphic: XY** using the **DEL** key!

Programming tool movements in Klartext

To program an NC block, initiate the dialog by pressing a function key. In the screen headline, the control then asks you for all the information necessary to program the desired function.



Example of a positioning block

L ▶ Press the **L** key

COORDINATES ?

X ▶ **10** (enter the target coordinate for the X axis)

Y ▶ **20** (enter the target coordinate for the Y axis)

ENT ▶ Go to the next question with **ENT**.

Radius comp.: RL/RR/no comp.?

ENT ▶ Enter **No radius compensation** and go to the next question with **ENT**

Feed rate F=? / F MAX = ENT

▶ **100** (enter a feed rate of 100 mm/min for this path contour)

ENT ▶ Go to the next question with **ENT**.

MISCELLANEOUS FUNCTION M ?

▶ **3** (enter the miscellaneous function **M3 Spindle on**)

END ▶ With the **END** key, the control ends this dialog.

Example

```
3 L  X+10 Y+5  R0 F100 M3
```

Possible feed rate input

Soft key	Functions for setting the feed rate
F MAX	Rapid traverse, blockwise. Exception: If defined before an **APPR** block, **FMAX** also in effect for moving to an auxiliary point **Further information:** "Important positions for approach and departure", Page 147
F AUTO	Traverse feed rate automatically calculated in **TOOL CALL**
F	Move at the programmed feed rate (unit of measure is mm/min or 1/10 inch/min). With rotary axes, the control interprets the feed rate in degrees/min, regardless of whether the NC program is written in mm or inches
FU	Define the feed per revolution (units in mm/1 or inch/1). Caution: In inch-programs, FU cannot be combined with M136
FZ	Define the tooth feed (units in mm/tooth or inch/tooth). The number of teeth must be defined in the tool table in the **CUT** column.

Key	Functions for conversational guidance
NO ENT	Ignore the dialog question
END	End the dialog immediately
DEL	Abort the dialog and erase the block

Actual position capture

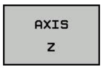
The control enables you to transfer the current tool position into the NC program,for example during

- Positioning-block programming
- Cycle programming

To transfer the correct position values, proceed as follows:

- ▶ Place the input box at the position in the NC block where you want to insert a position value

- ▶ Select the actual-position-capture function
- \> In the soft-key row the control displays the axes whose positions can be transferred.



- ▶ Select the axis
- \> The control writes the current position of the selected axis into the active input box.

In the working plane the control always captures the coordinates of the tool center, even though tool radius compensation is active.

The control takes the active tool length compensation into account and always captures the coordinate of the tool tip in the tool axis.

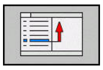
The control keeps the soft-key row for axis selection active until the **actual position capture** key is pressed again. This behavior remains in effect even if you save the current NC block or open a new NC block with a path function key. If you have to choose an input alternative via soft key (e.g. for radius compensation), then the control closes the soft-key row for axis selection.

The **Actual position capture** function is not allowed when the **Tilt working plane** function is active.

Editing an NC program

> **i** You cannot edit the active NC program while it is being run.

While you are creating or editing a NC program, you can select any desired line in the NC program or individual words in an NC block with the arrow keys or the soft keys:

Soft key/key	Function
	Change the position of the current NC block on the screen. Press this soft key to display additional NC blocks that are programmed before the current NC block No function if the NC program is fully visible on the screen
	Change the position of the current NC block on the screen. Press this soft key to display additional NC blocks that are programmed after the current NC block No function if the NC program is fully visible on the screen
 	Move from one NC block to the next NC block
 	Select individual words in an NC block
	Select a specific NC block **Further information:** "Using the GOTO key", Page 190

Soft key/key	Function
CE	■ Set the selected word to zero ■ Erase an incorrect number ■ Delete the (clearable) error message
NO ENT	Delete the selected word
DEL	■ Delete the selected NC block ■ Erase cycles and program sections
INSERT LAST NC BLOCK	Insert the NC block that you last edited or deleted

Inserting an NC block at any desired location

- Select the NC block after which you want to insert a new NC block
- Dialog initiation

Saving changes

The control normally saves changes automatically if you switch the operating mode or if you select the file manager. If you deliberately want to save changes to the NC program, proceed as follows:

- Select the soft-key row with the saving functions

STORE

- Press the **STORE** soft key
- > The control saves all changes made since the last time you saved the program.

Saving an NC program to a new file

You can save the contents of the currently active NC program under a different program name. Proceed as follows:

- Select the soft-key row with the saving functions

SAVE AS

- Press the **SAVE AS** soft key
- > The control opens a window in which you can enter the directory and the new file name.
- If necessary, select the target directory with the **SWITCH** soft key
- Enter the file name
- Confirm with the **OK** soft key or the **ENT** key, or abort the process by pressing the **CANCEL** soft key

> The file saved with **SAVE AS** can also be found in the file manager by pressing the **LAST FILES** soft key.

Undoing changes

You can undo all changes made since the last time you saved the program. Proceed as follows:

- ▶ Select the soft-key row with the saving functions

- ▶ Press the **CANCEL CHANGE** soft key
- > The control opens a window in which you can confirm or cancel this action.
- ▶ Reject the changes with the **YES** soft key or **ENT** key, or cancel the process with the **NO** soft key

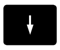
Editing and inserting words

- ▶ Select a word in an NC block
- ▶ Overwrite it with the new value
- > The dialog is available while the word is highlighted.
- ▶ To accept the change, press the **END** key

If you want to insert a word, press the horizontal arrow key repeatedly until the desired dialog appears. You can then enter the desired value.

Looking for the same words in different NC blocks

- ▶ Select a word in an NC block: Press the arrow key repeatedly until the desired word is highlighted



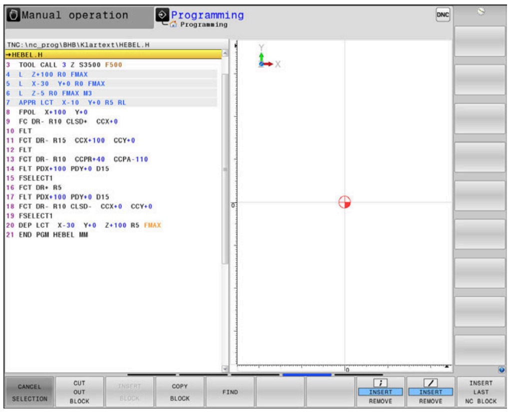
- ▶ Select an NC block with the arrow keys
  - Arrow down: search forwards
  - Arrow up: search backwards

The word that is highlighted in the new NC block is the same as the one you selected previously.

> **i** If you start a search in a very long NC program, the control shows a progress indicator. You can cancel the search at any time, if necessary.

Marking, copying, cutting and inserting program sections

The control provides the following functions for copying program sections within an NC program or into another NC program:



Soft key	Function
SELECT BLOCK	Switch the marking function on
CANCEL SELECTION	Switch the marking function off
CUT OUT BLOCK	Cut the marked block
INSERT BLOCK	Insert the block that is stored in the buffer memory
COPY BLOCK	Copy the marked block

To copy a program section, proceed as follows:

- ▶ Select the soft key row containing the marking functions
- ▶ Select the first NC block of the section you wish to copy
- ▶ Mark the first NC block: Press the **SELECT BLOCK** soft key.
- \> The control highlights the block in color and displays the **CANCEL SELECTION** soft key.
- ▶ Place the cursor on the last NC block of the program section you wish to copy or cut.
- \> The control shows the marked NC blocks in a different color. You can end the marking function at any time by pressing the **CANCEL SELECTION** soft key.
- ▶ Copy the selected program section: Press the **COPY BLOCK** soft key. Cut the selected program section: Press the **CUT OUT BLOCK** soft key.
- \> The control stores the selected block.

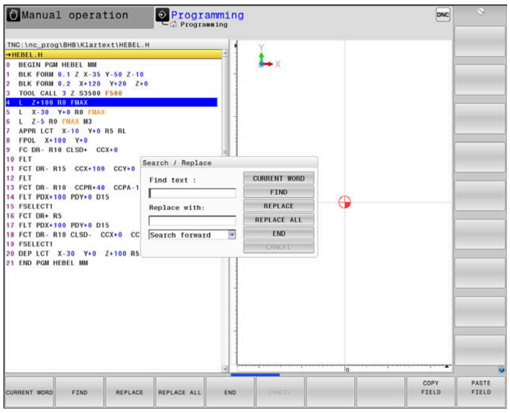
> If you want to transfer a program section to another NC program, you now need to select the desired NC program in the file manager.

- ▶ Use the arrow keys to select the NC block after which you want to insert the copied/cut section
- ▶ Insert the saved program section: Press the **INSERT BLOCK** soft key
- ▶ End the marking function: Press the **CANCEL SELECTION** soft key

The control's search function

The search function of the control enables you to search for any text within an NC program and replace it by a new text, if required.

Finding any text



FIND

- ▶ Select the search function
- > The control superimposes the search window and displays the available search functions in the soft-key row.
- ▶ Enter the text to be searched for, e.g.: **TOOL**
- ▶ Select forwards search or backwards search

FIND

- ▶ Start the search process
- > The control moves to the next NC block containing the text you are searching for

FIND

- ▶ Repeat the search process
- > The control moves to the next NC block containing the text you are searching for

END

- ▶ Terminate the search function: Press the END soft key

Finding/Replacing any text

NOTICE

Caution: Data may be lost!

The **REPLACE** and **REPLACE ALL** functions overwrite all of the found syntax elements without a confirmation prompt. The original file is not automatically backed up by the control before the replacement process. As a result, NC programs may be irreversibly damaged.

- Back up the NC programs, if required, before you start the replacement
- Be appropriately careful when using **REPLACE** and **REPLACE ALL**

The **FIND** and **REPLACE** functions cannot be used in the active NC program while the program is running. These functions are also not available if write protection is active.

- Select the NC block containing the word you wish to find

FIND

- Select the search function
- > The control superimposes the search window and displays the available search functions in the soft-key row.
- Press the **CURRENT WORD** soft key
- > The control loads the first word of the current NC block. If required, press the soft key again to load the desired word.

FIND

- Start the search process
- > The control moves to the next occurrence of the text you are searching for.

REPLACE

- To replace the text and then move to the next occurrence of the text, press the **REPLACE** soft key. Or, to replace all text occurrences, press the **REPLACE ALL** soft key. Or, to skip the text and move to its next occurrence, press the **FIND** soft key

END

- Terminate the search function: Press the END soft key

3.6 File management

Files

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

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

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

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

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

File names

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

File name	File type
PROG20	.H

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

The following characters are permitted:

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

The following characters have special meanings:

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

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

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

> **i** The maximum permitted path length is 255 characters. The path length consists of the drive characters, the directory name and the file name, including the extension.
>
> **Further information:** "Paths", Page 105

Displaying externally generated files on the control

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

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

Further information: User's Manual for Setup, Testing and Running NC Programs

Directories

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

Paths

A path indicates the drive and all directories and subdirectories under which a file is saved. The individual names are separated by a backslash ****.

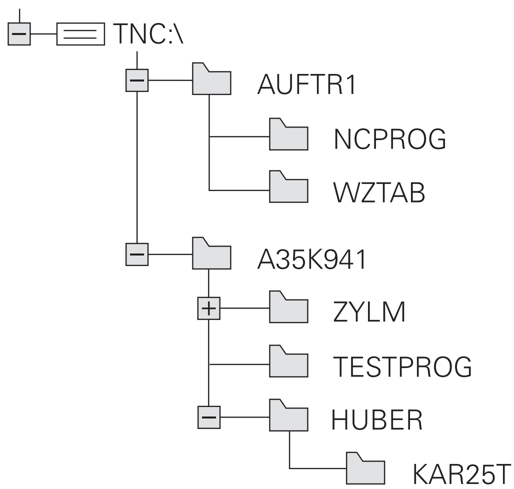
> The maximum permitted path length is 255 characters. The path length consists of the drive characters, the directory name and the file name, including the extension.

Example

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

TNC:\AUFTR1\NCPROG\PROG1.H

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



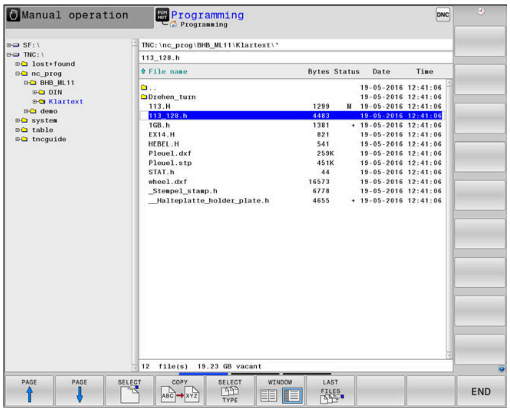
Overview: Functions of the file manager

Soft key	Function	Page
COPY ABC XYZ	Copy a single file	110
SELECT TYPE	Display a specific file type	108
NEW FILE	Create new file	110
LAST FILES	Display the last 10 files that were selected	114
DELETE	Delete a file	115
TAG	Tag a file	116
RENAME ABC = XYZ	Rename file	117
PROTECT	Protect a file against editing and erasure	118
UNPROTECT	Cancel file protection	118
ADAPT NC PGM / TABLE	Import file of an iTNC 530	See the User's Manual for Setup, Testing and Running NC Programs
	Customize table view	391
NET	Manage network drives	See the User's Manual for Setup, Testing and Running NC Programs
SELECT EDITOR	Select the editor	118
SORT	Sort files by properties	117
COPY DIR	Copy a directory	114
DELETE ALL	Delete directory with all its subdirectories	
UPDATE TREE	Refresh directory	
RENAME ABC = XYZ	Rename a directory	
NEW DIRECTORY	Create a new directory	

Calling the file manager

PGM MGT

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



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

If the directory tree is longer than the screen, navigate using the scroll bar or a connected mouse.

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

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

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

Selecting drives, directories and files

PGM MGT

- Call the file manager by pressing the **PGM MGT** key

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

- Moves the cursor from the left to the right window, and vice versa
- Moves the cursor up and down within a window

PAGE PAGE

- Moves the cursor one page up or down within a window

Step 1: Select drive

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

SELECT

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

ENT

- Press the **ENT** key

Step 2: Select a directory

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

Step 3: Select a file

SELECT TYPE

- Press the **SELECT TYPE** soft key

SHOW ALL

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

SELECT

- Press the **SELECT** soft key, or

ENT

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

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

Filtering the display

To filter the displayed files, proceed as follows:

SELECT TYPE

- Press the **SELECT TYPE** soft key

SHOW .H

- Press the soft key for the desired file type

Alternative:

SHOW ALL

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

Alternative:

DISPLAY FILTER

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

Alternative:

DISPLAY FILTER

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

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

Creating a new directory

- Move the highlight in the left window to the directory in which you want to create a subdirectory
  - Press the **NEW DIRECTORY** soft key
  - Enter a directory name
  - Press the **ENT** key
  - Press the **OK** soft key to confirm or
  - Press the **CANCEL** soft key to abort

Creating new file

- Select the directory in the left window in which you wish to create the new file
- Position the cursor in the right window
  - Press the **NEW FILE** soft key
  - Enter the file name with extension
  - Press the **ENT** key

Copying a single file

- Move the cursor to the file you wish to copy
  - Press the **COPY** soft key to select the copying function
  - > The control opens a pop-up window.

Copying files into the current directory

- Enter the name of the destination file.
- Press the **ENT** key or the **OK** soft key
- > The control copies the file to the active directory. The original file is retained.

Copying files into another directory

- Press the **Target Directory** soft key to select the target directory from a pop-up window
- Press the **ENT** key or the **OK** soft key
- > The control copies the file under the same name to the selected directory. The original file is retained.

> **i** When you start the copying process with the **ENT** key or the **OK** soft key, the control displays a pop-up window with a progress indicator.

Copying files into another directory

- Select a screen layout with two equally sized windows

In the right window

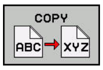
- Press the **SHOW TREE** soft key
- Move the cursor to the directory into which you wish to copy the files, and display the files in this directory with the **ENT** key

In the left window

- Press the **SHOW TREE** soft key
- Select the directory with the files that you want to copy, and display the files with the **SHOW FILES** soft key

- Press the Tag soft key: Call the file tagging functions

- Press the Tag soft key: Position the cursor on the file you wish to copy and tag. You can tag several files in this way, if desired



- Press the Copy soft key: Copy the tagged files into the target directory

Further information: "Tagging files", Page 116

If you have tagged files in both the left and right windows, the control copies from the directory in which the cursor is located.

Overwriting files

If you copy files to a directory in which other files are stored under the same name, the control will ask whether the files in the target directory should be overwritten:

- Overwrite all files (**Existing files** field selected): Press the **OK** soft key, or
- To leave the files as they are, press the **CANCEL** soft key

If you want to overwrite a protected file, select the **Protected files** field, or cancel the process.

Copying a table

Importing lines to a table

If you are copying a table into an existing table, you can overwrite individual lines with the **REPLACE FIELDS** soft key. Prerequisites:

- The target table must exist
- The file to be copied must only contain the lines you want to replace
- Both tables must have the same file extension

NOTICE

Caution: Data may be lost!

The **REPLACE FIELDS** function overwrites all lines of the target file that are contained in the copied table without a confirmation prompt. The original file is not automatically backed up by the control before the replacement process. As a consequence, tables may be irreversibly damaged.

- ▶ Back up the tables, if required, before you start the replacement
- ▶ Be accordingly careful when using **REPLACE FIELDS**

Example

With a tool presetter you have measured the length and radius of ten new tools. The tool presetter then generates the TOOL_Import.T tool table with 10 lines (for the 10 tools).

Proceed as follows:

- ▶ Copy this table from the external data medium to any directory
- ▶ Copy the externally created table to the existing table TOOL.T using the control's file manager.
- > The control asks you whether you want to overwrite the existing TOOL.T tool table.
- ▶ Press the **YES** soft key
- > The control will completely overwrite the current TOOL.T tool table. After this copying process the new TOOL.T table consists of 10 lines.
- ▶ Alternative: Press the **REPLACE FIELDS** soft key
- > The control overwrites the 10 lines in the TOOL.T file. The data of the other lines is not changed.

Extracting lines from a table

You can select one or more lines in a table and save them in a separate table.

Proceed as follows:

- Open the table from which you want to copy lines
- Use the arrow keys to select the first line to be copied
- Press the **MORE FUNCTIONS** soft key
- Press the **TAG** soft key
- Select additional lines, if required
- Press the **SAVE AS** soft key
- Enter a name for the table in which the selected lines are to be saved

Copying a directory

- Move the highlight in the right window onto the directory you want to copy
- Press the **COPY** soft key
- > The control opens the window for selecting the target directory.
- Select the target directory and confirm with the **ENT** key or the **OK** soft key
- > The control copies the selected directory and all its subdirectories to the selected target directory.

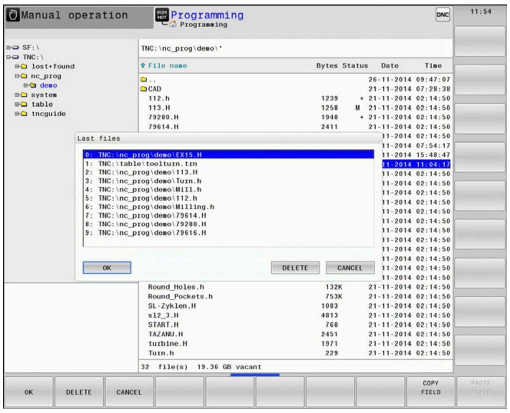
Choosing one of the last files selected

PGM MGT

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

LAST FILES

- Display the last ten files selected: Press the **LAST FILES** soft key



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

- Moves the cursor up and down within a window

OK

- Select the file: Press the **OK** soft key, or

ENT

- Press the **ENT** key

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

Deleting a file

NOTICE

Caution: Data may be lost!

The **DELETE** function permanently deletes the file. The control does not perform an automatic backup of the file prior to deletion (e.g., there is no recycle bin). Files are thereby irreversibly deleted.

- Regularly back up important data to external drives

Proceed as follows:

- Move the cursor to the file you want to delete

- Press the **DELETE** soft key
- > The control asks whether you want to delete the file.
- Press the **OK** soft key
- > The control deletes the file.
- Alternative: Press the **CANCEL** soft key
- > The control aborts the procedure.

Deleting a directory

NOTICE

Caution: Data may be lost!

The **DELETE ALL** function permanently deletes all files of the directory. The control does not perform an automatic backup of the files prior to deletion (e.g., there is no recycle bin). Files are thereby irreversibly deleted.

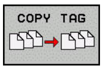
- Regularly back up important data to external drives

Proceed as follows:

- Move the cursor to the directory you want to delete

- Press the **DELETE ALL** soft key
- > The control inquires whether you really intend to delete the directory and all its subdirectories and files.
- Press the **OK** soft key
- > The control deletes the directory.
- Alternative: Press the **CANCEL** soft key
- > The control aborts the procedure.

Tagging files

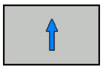
Soft key	Tagging function
	Tag a single file
	Tag all files in the directory
	Untag a single file
	Untag all files
	Copy all tagged files

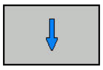
Some functions, such as copying or erasing files, can not only be used for individual files, but also for several files at once. To tag several files, proceed as follows:

- Move the cursor to the first file

- To display the tagging functions, press the **TAG** soft key

- To tag the file, press the **TAG FILE** soft key

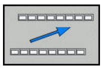




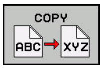
- Move the cursor to other files

- To select a another file, press the **TAG FILE** soft key, etc.

To copy tagged files:

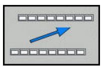


- Leave the active soft-key row



- Press the **COPY** soft key

To delete tagged files:

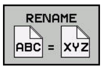


- Leave the active soft-key row

- Press the **DELETE** soft key

Renaming a file

- Move the cursor to the file you wish to rename



- Select the function for renaming: Press the **RENAME** soft key
- Enter the new file name; the file type cannot be changed
- To rename: Press the **OK** soft key or the **ENT** key

Sorting files

- Select the folder in which you wish to sort the files

- Press the **SORT** soft key
- Select the soft key with the corresponding display criterion
  - **SORT BY NAME**
  - **SORT BY SIZE**
  - **SORT BY DATE**
  - **SORT BY TYPE**
  - **SORT BY STATUS**
  - **UNSORTED**

Additional functions

Protecting a file and canceling file protection

- Place the cursor on the file you want to protect
- Select the additional functions: Press the **MORE FUNCTIONS** soft key
- Activate file protection: Press the **PROTECT** soft key
- > The file is tagged with the "protected" symbol.
- Cancel file protection: Press the **UNPROTECT** soft key

Selecting the editor

- Place the cursor on the file you want to open
- Select the additional functions: Press the **MORE FUNCTIONS** soft key
- Select the editor: Press the **SELECT EDITOR** soft key
- Mark the desired editor
  - **TEXT EDITOR** for text files, e.g. **.A** or **.TXT**
  - **PROGRAM EDITOR** for NC programs **.H** and **.I**
  - **TABLE EDITOR** for tables, e.g. **.TAB** or **.T**
  - **BPM EDITOR** for pallet tables **.P**
- Press the **OK** soft key

Connecting and removing USB storage devices

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

To remove a USB device, proceed as follows:

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

Further information: User's Manual for Setup, Testing and Running NC Programs

ADVANCED ACCESS RIGHTS

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

Further information: User's Manual for Setup, Testing and Running NC Programs

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

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

Further information: User's Manual for Setup, Testing and Running NC Programs

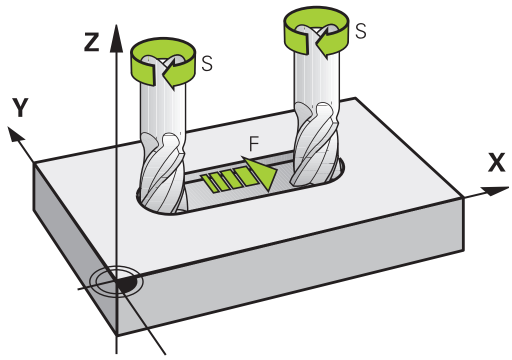
4

Tools

4.1 Entering tool-related data

Feed rate F

The feed rate **F** is the speed at which the tool center point moves. The maximum feed rates can be different for the individual axes and are set in machine parameters.



Input

You can enter the feed rate in the **TOOL CALL** block and in every positioning block.

Further information: "Creating the NC blocks with the path function keys ", Page 142

You enter the feed rate **F** in mm/min in millimeter programs, and in 1/10 inch/min in inch-programs, for resolution reasons. Alternatively, with the corresponding soft keys, you can also define the feed rate in mm per revolution (mm/1) **FU** or in mm per tooth (mm/tooth) **FZ**.

Rapid traverse

If you wish to program rapid traverse, enter **F MAX.** To enter **FMAX,** press the **ENT** key or the **FMAX** soft key when the dialog question **FEED RATE F = ?** appears on the control's screen.

> To move your machine at rapid traverse, you can also program the corresponding numerical value, e.g. **F30000**. Unlike **FMAX**, this rapid traverse remains in effect not only in the individual block but in all blocks until you program a new feed rate.

Duration of effect

A feed rate entered as a numerical value remains in effect until an NC block with a different feed rate is reached. **FMAX** is only effective in the NC block in which it is programmed. After the NC block with **F MAX** is executed, the feed rate will return to the last feed rate entered as a numerical value.

Changing during program run

You can adjust the feed rate during the program run with the feed rate potentiometer F.

The feed rate potentiometer lowers the programmed feed rate, not the feed rate calculated by the control.

Spindle speed S

The spindle speed S is entered in revolutions per minute (rpm) in a **TOOL CALL** block (tool call). Instead, you can also define the cutting speed Vc in meters per minute (m/min).

Programmed change

In the NC program, you can change the spindle speed in a **TOOL CALL** block by entering only the new spindle speed.

Proceed as follows:

TOOL CALL

- Press the **TOOL CALL** key
- Ignore the dialog question for **Tool number ?** with the **NO ENT** key
- Ignore the dialog question for **Working spindle axis X/Y/Z ?** with the **NO ENT** key
- Enter the new spindle speed at the **Spindle speed S= ?** prompt, or switch to entry of the cutting speed by pressing the **VC** soft key

END

- Confirm your input with the **END** key

In the following cases the control changes only the speed:

- **TOOL CALL** block without tool name, tool number, and tool axis
- **TOOL CALL** block without tool name, tool number, with the same tool axis as in the previous **TOOL CALL** block

In the following cases the control runs the tool-change macro and inserts a replacement tool if necessary:

- **TOOL CALL** block with tool number
- **TOOL CALL** block with tool name
- **TOOL CALL** block without tool name or tool number, with a changed tool axis direction

Changing during program run

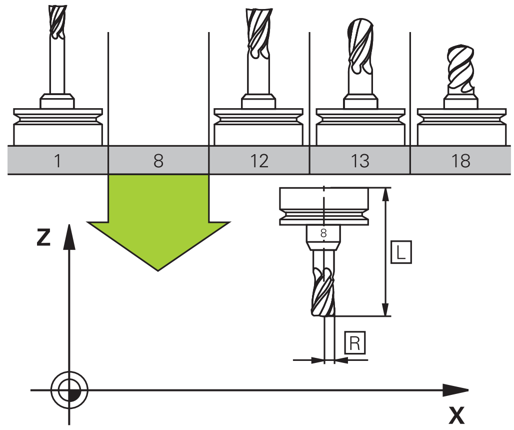
You can adjust the spindle speed during program run with the spindle speed potentiometer S.

4.2 Tool data

Requirements for tool compensation

You usually program the coordinates of path contours as they are dimensioned in the workpiece drawing. To allow the control to calculate the tool center path (i.e. the tool compensation) you must also enter the length and radius of each tool you are using.

You can enter tool data either directly in the NC program with **TOOL DEF** or separately in a tool tables. In a tool table, you can also enter additional data for the specific tool. The control will consider all the data entered for the tool when executing the NC program.



Tool number, tool name

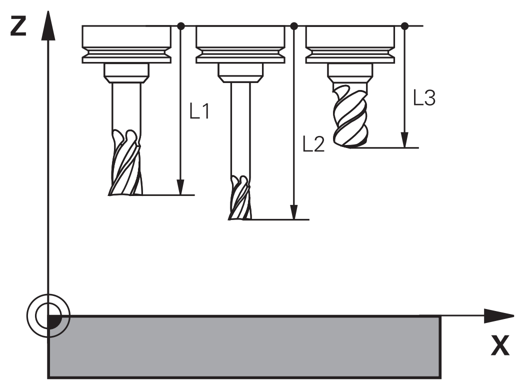
Each tool is identified by a number between 0 and 32767. If you are working with tool tables, you can also enter a tool name for each tool. Tool names can have up to 32 characters.

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

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

Tool length L

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



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

Measuring the tool length

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

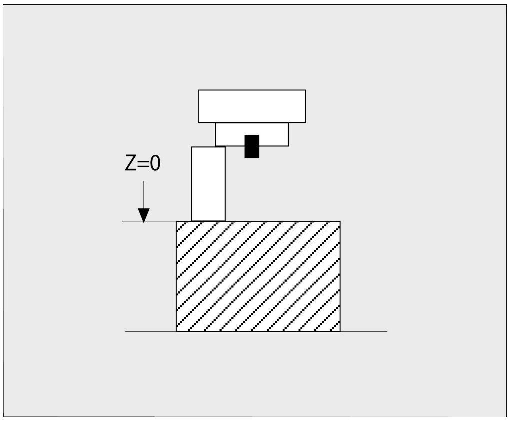
You have the following options for determining the tool length:

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

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

Determining the tool length with a gauge block

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



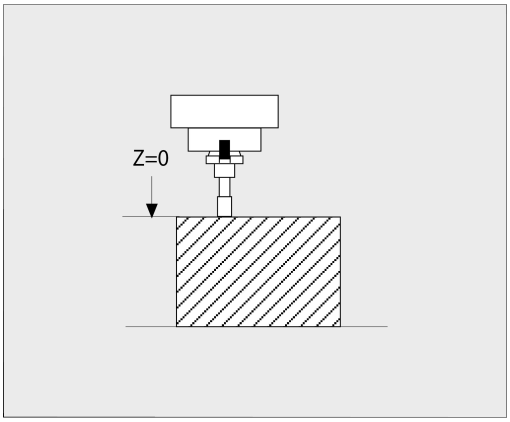
Proceed as follows to set the datum with a gauge block:

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

To determine the tool length, proceed as follows:

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

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



Proceed as follows to set the preset with a calibration pin and a tool setter:

- ▶ Clamp the tool setter onto the machine table.
- ▶ Bring the flexible inner ring of the tool setter to the same height as the fixed outer ring.
- ▶ Set the gauge to 0
- ▶ Move the calibration pin onto the flexible inner ring.
- ▶ Set the datum in **Z**

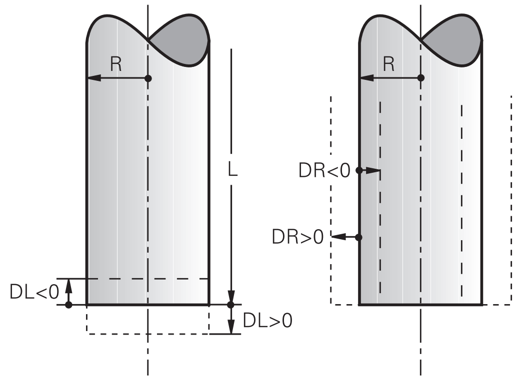
To determine the tool length, proceed as follows:

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

Tool radius R

You can enter the tool radius R directly.

Delta values for lengths and radii



Delta values are offsets in the length and radius of a tool.

A positive delta value represents a tool oversize (**DL**, **DR**>0). For a machining operation with an oversize, enter the value for the oversize in the NC program with **TOOL CALL** or with the help of a compensation table.

A negative delta value describes a tool undersize (**DL**, **DR**<0). An undersize is entered in the tool table for wear.

Delta values are usually entered as numerical values. In a **TOOL CALL** block, you can also assign the values to Q parameters.

Input range: You can enter a delta value with up to ± 99.999 mm.

> Delta values from the tool table influence the graphical representation of the clearing simulation.
>
> Delta values from the NC program do not change the depicted size of the **tool** in the simulation. However, the programmed delta values move the **tool** in the simulation by the amount of the defined value.

> Delta values from the **TOOL CALL** block influence the position display depending on the optional machine parameter **progToolCallDL** (no. 124501).

Entering tool data into the NC program

> Refer to your machine manual!
>
> The machine tool builder determines the scope of functions of the **TOOL DEF** function.

The number, length, and radius of a specific tool are defined in the **TOOL DEF** block of the NC program:

Proceed as follows for the definition:

TOOL DEF

- press the **TOOL DEF** key.

TOOL NUMBER

- Press the appropriate soft key
  - **Tool number**
  - **TOOL NAME**
  - **QS**
- **Tool length**: Compensation value for the tool length
- **Tool radius**: Compensation value for the tool radius

Example

```
4 TOOL DEF  5 L+10  R+5
```

Calling the tool data

Before you can call the tool, you have to define it in a **TOOL DEF** block or in the tool table.

A **TOOL CALL** in the NC program is programmed with the following data:

- Press the **TOOL CALL** key
- **Tool number**: Enter the number or name of the tool. With the **TOOL NAME** soft key, you can enter a name. With the **QS** soft key, you enter a string parameter. The control automatically puts the tool name in quotation marks. You must first assign a tool name to a string parameter. The names refer to an entry in the active tool table TOOL .T.

- Alternative: Press the **SELECT** soft key
- > The control opens a window where you can select a tool directly from the TOOL.T tool table.
- To call a tool with other compensation values, enter a decimal point followed by the index you defined in the tool table.
- **Working spindle axis X/Y/Z**: Enter the tool axis
- **Spindle speed S**: Enter the spindle speed S in revolutions per minute (rpm) Alternatively, you can define the cutting speed Vc in meters per minute (m/min). Press the **VC** soft key
- **Feed rate F**: Enter feed rate **F** in millimeters per minute (mm/min). Alternatively, you can define the feed rate in millimeters per revolution (mm/1) by pressing the **FU** soft key or in millimeters per tooth (mm/tooth) by pressing **FZ**. The feed rate is effective until you program a new feed rate in a positioning block or in a **TOOL CALL** block
- **Tool length oversize DL**: Enter the delta value for the tool length
- **Tool radius oversize DR**: Enter the delta value for the tool radius
- **Tool radius oversize DR2**: Enter the delta value for the tool radius 2

> In the following cases the control changes only the speed:
> - **TOOL CALL** block without tool name, tool number, and tool axis
> - **TOOL CALL** block without tool name, tool number, with the same tool axis as in the previous **TOOL CALL** block
>
> In the following cases the control runs the tool-change macro and inserts a replacement tool if necessary:
> - **TOOL CALL** block with tool number
> - **TOOL CALL** block with tool name
> - **TOOL CALL** block without tool name or tool number, with a changed tool axis direction

Tool selection in the pop-up window

If you open a pop-up window for tool selection, the control marks all tools available in the tool magazine green.

You can search for a tool in the pop-up window:

GOTO □

- ▶ Press the **GOTO** key
- ▶ Alternative: Press the **FIND** soft key
- ▶ Enter the tool name or tool number

ENT

- ▶ Press the **ENT** key
- > The control goes to the first tool that matches the entered search string.

The following functions can be used with a connected mouse:

- You can sort the data in ascending or descending order by clicking a column of the table head.
- You can arrange the columns in any sequence you want by clicking a column of the table head and then moving it with the mouse key pressed down

The pop-up windows displayed for a tool number search and a tool name search can be configured separately. The sort order and the column widths are retained when the control is switched off.

Tool call

Call tool number 5 in the tool axis Z with a spindle speed 2500 rpm and a feed rate of 350 mm/min. The tool length and tool radius 2 are to be programmed with an oversize of 0.2 and 0.05 mm, the tool radius with an undersize of 1 mm.

Example

```
20 TOOL CALL  5.2 Z  S2500 F350  DL+0.2 DR-1  DR2+0.05
```

The character **D** preceding **L**, **R** and **R2** designates delta values.

Preselection of tools

> Refer to your machine manual!
> The preselection of tools with **TOOL DEF** can vary depending on the individual machine tool.

If you are working with tool tables, use a **TOOL DEF** block to preselect the next tool. Simply enter the tool number, or a Q or QS parameter, or type the tool name in quotation marks.

Tool change

Automatic tool change

Refer to your machine manual!

The tool change function can vary depending on the individual machine tool.

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

Automatic tool change if the tool life expires: M101

Refer to your machine manual!

The function of **M101** can vary depending on the individual machine tool.

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

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

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

NOTICE

Danger of collision!

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

- Deactivate the tool change with **M102**

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

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

Input parameter BT (block tolerance)

Depending on the NC program, the machining time can increase as a result of the tool life verification and calculation of the automatic tool change. You can influence this with the optional input parameter **BT** (block tolerance).

If you enter the **M101** function, the control continues the dialog by requesting **BT**. Here you define the number of NC blocks (1 to 100) by which the automatic tool change may be delayed. The resulting time period by which the tool change is delayed depends on the content of the NC blocks (e.g. feed rate, path). If you do not define **BT**, the control uses the value 1 or, if applicable, a default value defined by the machine manufacturer.

> The higher the value of **BT**, the smaller will be the effect of an extended program duration through the **M101** function. Please note that this will delay the automatic tool change!
>
> Use the formula **BT = 10: Average machining time of an NC block in seconds** to calculate a suitable starting value for **BT**. Round the result up to an integer value. If the calculated result is greater than 100, use the maximum input value of 100.
>
> If you want to reset the current age of a tool (e.g. after changing the indexable inserts), enter the value 0 in the CUR_TIME column.
>
> The **M101** function is not available for turning tools and in turning mode.

Prerequisites for a tool change with M101

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

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

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

Overtime for tool life

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

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

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

Prerequisites for NC blocks with surface-normal vectors and 3-D compensation

The active radius (**R** + **DR**) of the replacement tool must not deviate from the radius of the original tool. You can enter the delta values (**DR**) either in the tool table or in the NC program (compensation table or **TOOL CALL** block). If deviations occur, the control displays a message and does not replace the tool. You can suppress this message with the M function **M107,** and reactivate it with **M108.**

Further information: "Three-dimensional tool compensation (option 9)", Page 449

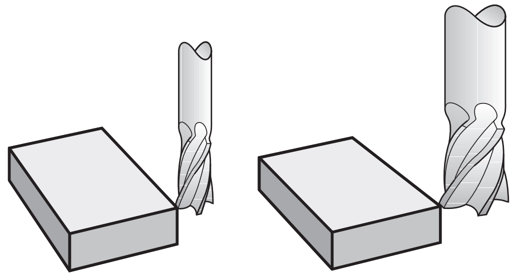
4.3 Tool compensation

Introduction

The control adjusts the tool path by the compensation value for the tool length in the spindle axis. In the machining plane, it compensates the tool radius.

If you are writing the NC program directly on the control, the tool radius compensation is effective only in the working plane.

The control accounts for the compensation value in up to six axes including the rotary axes.



Tool length compensation

Length compensation becomes effective automatically as soon as a tool is called. To cancel length compensation, call a tool with the length L=0 (e.g. **TOOL CALL 0**).

NOTICE

Danger of collision!

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

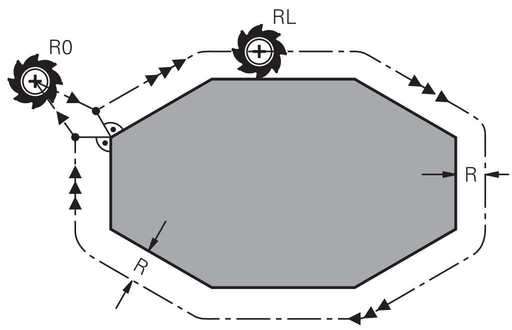
- Always define the actual tool length of a tool (not just the difference)
- Use **TOOL CALL 0** only to empty the spindle

With length compensation, delta values from both the NC program and the tool table are considered.

Compensation value = **L** + $\mathbf{DL}_{TAB}$ + $\mathbf{DL}_{Prog}$ with

L:	Tool length **L** from **TOOL DEF** block or tool table
$\mathbf{DL}_{TAB}$:	Oversize for length **DL** in the tool table
$\mathbf{DL}_{Prog}$:	Oversize **DL** for length from **TOOL CALL** block or from the compensation table The most recently programmed value takes effect. **Further information:** "Compensation table", Page 377

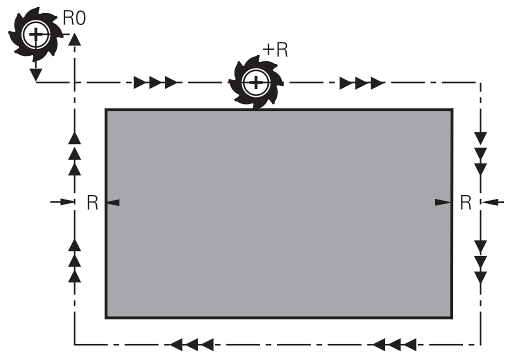
Tool radius compensation



An NC block can contain the following types of tool radius compensation:

- **RL** or **RR** for radius compensation of any contouring function
- **R0**, if there is no radius compensation
- **R+** lengthens a paraxial movement by the amount of the tool radius
- **R-** shortens an axis-parallel movement by the amount of the tool radius

> The control shows an active tool compensation in the general status display.



The radius compensation takes effect as soon as a tool is called and is moved with one of the abovementioned types of tool radius compensation within a straight-line block or within a paraxial movement in the working plane.

> The control automatically cancels radius compensation in the following cases:
> - Straight-line block with **R0**
> - **DEP** function for departing from the contour
> - Selection of a new NC program via **PGM MGT**

For radius compensation, the control takes the delta values from both the **TOOL CALL** block and the tool table into account:

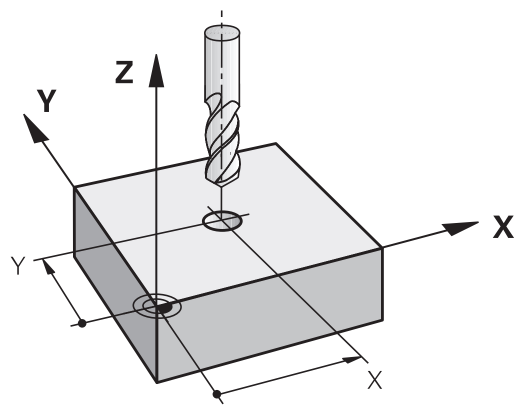
Compensation value = **R** + $\mathbf{DR}_{TAB}$ + $\mathbf{DR}_{Prog}$ with

R:	Tool radius **R** from **TOOL DEF** block or tool table
$\mathbf{DR}_{TAB}$:	Oversize for radius **DR** in the tool table
$\mathbf{DR}_{Prog}$:	Oversize **DR** for radius from **TOOL CALL** block or from the compensation table **Further information:** "Compensation table", Page 377

Movements without radius compensation: R0

The tool center moves in the working plane to the programmed coordinate.

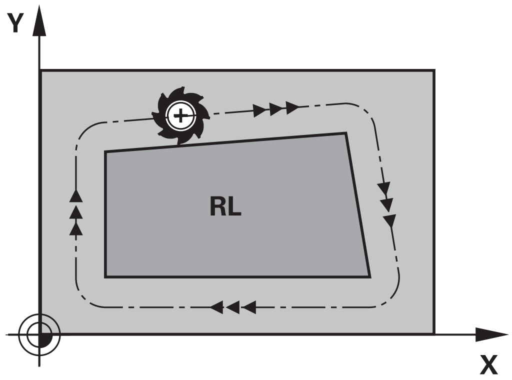
Applications: Drilling and boring, pre-positioning

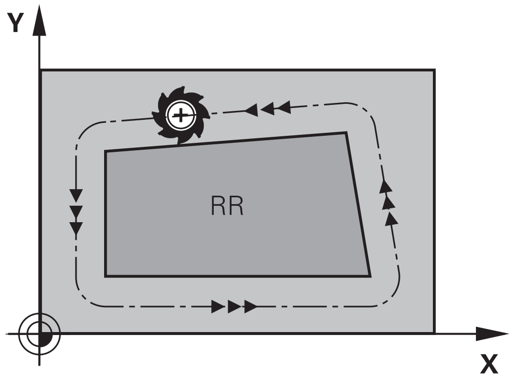


Contouring with radius compensation: RR and RL

RR: The tool moves to the right of the programmed contour

RL: The tool moves to the left of the programmed contour





The tool center moves along the contour at a distance equal to the radius. **Right** or **left** are to be understood as based on the direction of tool movement along the workpiece contour.

> Between two NC blocks with different radius compensations **RR** and **RL** you must program at least one traversing block in the working plane without radius compensation (that is, with **R0**).
>
> The control does not put radius compensation into effect until the end of the NC block in which it is first programmed.
>
> When radius compensation is activated with **RR/RL**, and in the case of cancellation with **R0**, the control always positions the tool perpendicularly to the programmed start or end point. Position the tool before the first contour point or after the last contour point such that the contour does not incur damage.

Input of radius compensation within path contours

Radius compensation is entered in an **L** block. Enter the coordinates of the target point and confirm your entry with the **ENT** key.

Radius comp.: RL/RR/no comp.?

RL

- Select tool movement to the left of the contour: Press the **RL** soft key, or

RR

- Select tool movement to the right of the contour: Press the **RR** soft key, or

ENT

- Select tool movement without radius compensation or cancel radius compensation: Press the **ENT** key

END

- Terminate the NC block: Press the **END** key

Input of radius compensation within paraxial movements

Radius compensation is entered in a positioning block. Enter the coordinates of the target point, and confirm your entry with the **ENT** key.

TOOL RADIUS COMP: R+/R-/NO COMP?

R+
- ▶ The TNC lengthens the traverse path of the tool by the amount of the tool radius

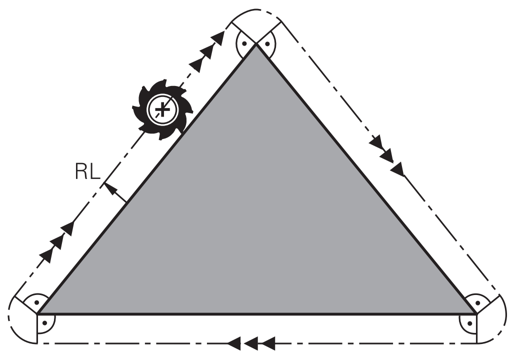
R-
- ▶ The TNC shortens the traverse path of the tool by the amount of the tool radius

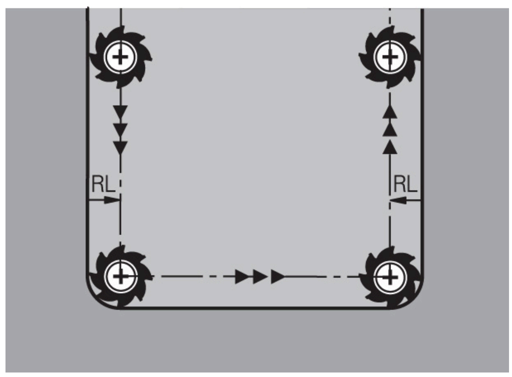
ENT
- ▶ Select tool movement without radius compensation, or cancel radius compensation: Press the **ENT** key

END
- ▶ Terminate the NC block: Press the **END** key

Radius compensation: Machining corners

- Outside corners:
  If you program radius compensation, the control moves the tool around outside corners on a transitional arc. If necessary, the control reduces the feed rate at outside corners to reduce machine stress, for example at very great changes of direction
- Inside corners:
  The control calculates the intersection of the tool center paths at inside corners under radius compensation. From this point it then starts the next contour element. This prevents damage to the workpiece at the inside corners. The permissible tool radius, therefore, is limited by the geometry of the programmed contour.





NOTICE

Danger of collision!

The control needs safe positions for contour approach and departure. These positions must enable the control to perform compensating movements when radius compensation is activated and deactivated. Incorrect positions can lead to contour damage. Danger of collision during machining!

- ▶ Program safe approach and departure positions at a sufficient distance from the contour
- ▶ Consider the tool radius
- ▶ Consider the approach strategy

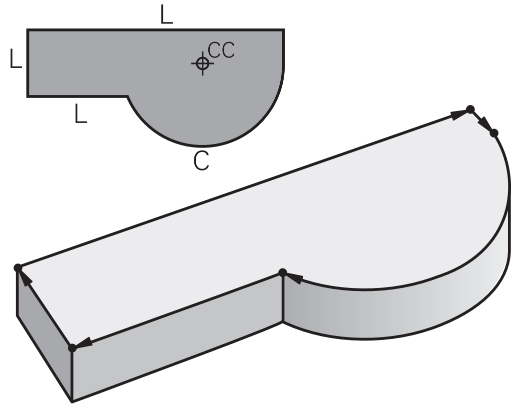
5

Programming Contours

5.1 Tool movements

Path functions

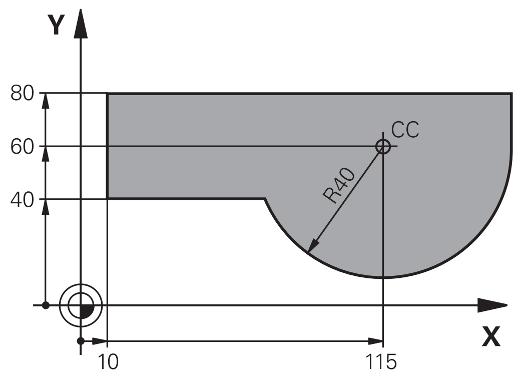
A workpiece contour is usually composed of several contour elements such as straight lines and circular arcs. With the path functions, you can program the tool movements for **straight lines** and **circular arcs**.



FK free contour programming

If a production drawing is not dimensioned for NC and the dimensions given are not sufficient for creating a part program, you can program the workpiece contour with the FK free contour programming. The control calculates the missing data.

With FK programming, you also program tool movements for **straight lines** and **circular arcs**.



Miscellaneous functions M

With the control's miscellaneous functions you can affect

- the program run, e.g., a program interruption
- the machine functions, such as switching spindle rotation and coolant supply on and off
- the path behavior of the tool

Subprograms and program section repeats

If a machining sequence occurs several times in a program, you can save time and reduce the chance of programming errors by entering the sequence once and then defining it as a subprogram or program-section repeat. If you wish to execute a specific NC program section only under certain conditions, you also define this machining sequence as a subprogram. In addition, you can have an NC program call a separate NC program for execution.

Further information: "Subprograms and Program Section Repeats", Page 243

Programming with Q parameters

Instead of programming numerical values in an NC program, you enter markers called Q parameters. You can use the Q parameters for programming mathematical functions that control program execution or describe a contour.

In addition, programming with Q parameters enables you to measure with the 3-D touch probe during the program run.

Further information: "Programming Q Parameters", Page 263

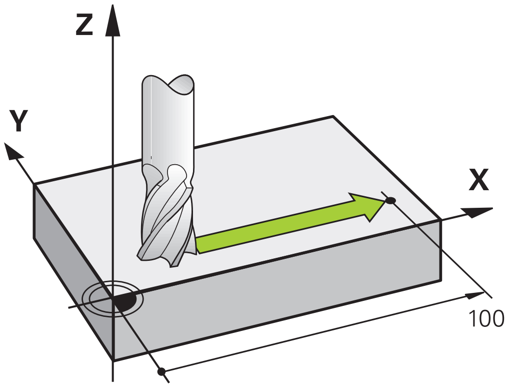
5.2 Fundamentals of path functions

Programming tool movements for machining

You create an NC program by programming the path functions for the individual contour elements in sequence You do this by entering the coordinates of the end points of the contour elements given in the production drawing. The control calculates the actual path of the tool from these coordinates, and from the tool data and radius compensation.

The control moves all machine axes programmed in the NC block of a path function simultaneously.

Movement parallel to the machine axes



If the NC block contains one coordinate, the control moves the tool parallel to the programmed machine axis.

Depending on the individual machine, the machining program is executed by movement of either the tool or the machine table on which the workpiece is clamped. Path contours are programmed as if the tool were moving.

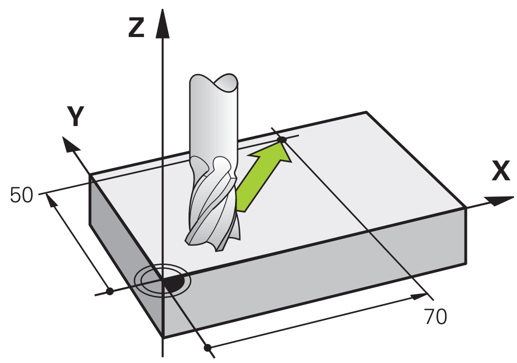
Example

```
50 L  X+100
```

50	Block number
L	Path function **straight line**
X+100	Coordinate of the end point

The tool retains the Y and Z coordinates and moves to the position X=100.

Movement in the main planes



If the NC block contains two coordinates, the control moves the tool in the programmed plane.

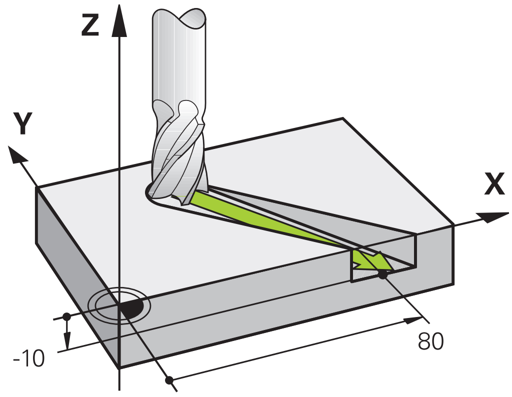
Example

```
L X+70 Y+50
```

The tool retains the Z coordinate and moves on the XY plane to the position X=70, Y=50.

Three-dimensional movement

If the NC block contains three coordinates, the control moves the tool spatially to the programmed position.



Example

```
L X+80  Y+0 Z-10
```

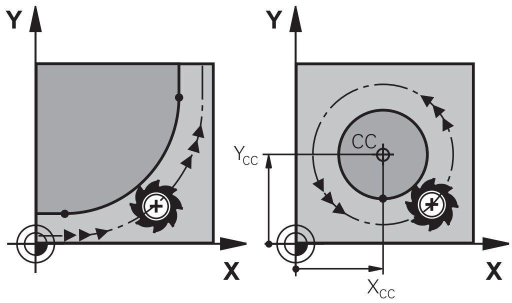
You can program up to six axes in a straight line block according to the kinematics of your machine.

Example

```
L X+80 Y+0 Z-10 A+15 B+0 C-45
```

Circles and circular arcs

The control moves two machine axes simultaneously on a circular path relative to the workpiece. You can define a circular movement by entering the circle center **CC** .



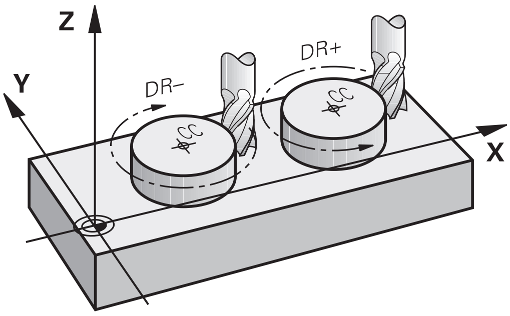
When you program a circle, the control assigns it to one of the main planes. This plane is defined automatically when you set the spindle axis during a **TOOL CALL**:

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

> You can program circles that do not lie parallel to a main plane by using the function for **Tilt working plane** or with Q parameters.
>
> **Further information:** "The PLANE function: Tilting the working plane (option 8)", Page 403
>
> **Further information:** "Principle and overview of functions", Page 264

Direction of rotation DR for circular movements

When a circular path has no tangential transition to another contour element, enter the direction of rotation as follows:



Clockwise direction of rotation: **DR-**

Counterclockwise direction of rotation: **DR+**

Radius compensation

The radius compensation must be in the NC block in which you move to the first contour element. You cannot activate radius compensation in an NC block for a circular path. It must be activated beforehand in a straight-line block.

Further information: "Path contours — Cartesian coordinates", Page 154

Further information: "Approaching and departing a contour", Page 144

Pre-positioning

NOTICE

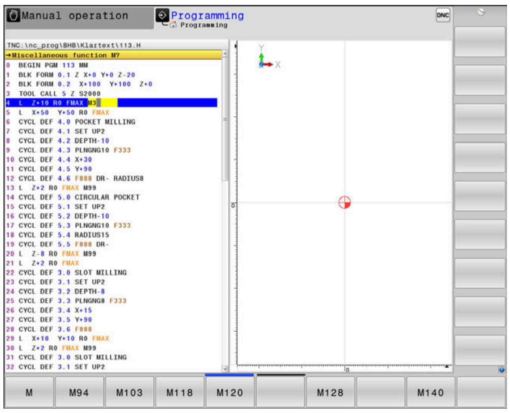
Danger of collision!

The control does not automatically check whether collisions can occur between the tool and the workpiece. Incorrect pre-positioning can also lead to contour damage. There is danger of collision during the approach movement!

- Program a suitable pre-position
- Check the sequence and contour with the aid of the graphic simulation

Creating the NC blocks with the path function keys

The gray path function keys initiate the dialog. The control asks you successively for all the necessary information and inserts the program block into the NC program.



Example – programming a straight line

L

- Initiate the programming dialog, e.g. for a straight line

COORDINATES?

X

- Enter the coordinates of the straight-line end point, e.g. -20 in X

COORDINATES?

Y

- Enter the coordinates of the straight-line end point, e.g. 30 in Y, and confirm with the **ENT** key

Radius comp.: RL/RR/no comp.?

R0

- Select the radius compensation (here, press the **R0** soft key—the tool moves without compensation)

Feed rate F=? / F MAX = ENT

ENT

- Enter **100** (feed rate e.g. 100 mm/min; for programming in inches: an input of 100 corresponds to a feed rate of 10 inches/min) and confirm your entry with the **ENT** key, or

F MAX

- Move at rapid traverse: Press the **FMAX** soft key, or

F AUTO

- Traverse with the feed rate defined in the **TOOL CALL** block: Press the **F AUTO** soft key.

MISCELLANEOUS FUNCTION M?

END

- Enter **3** (miscellaneous function e.g. M3) and terminate the dialog with the **END** key

Example

```
L X-20 Y+30 R0 FMAX M3
```

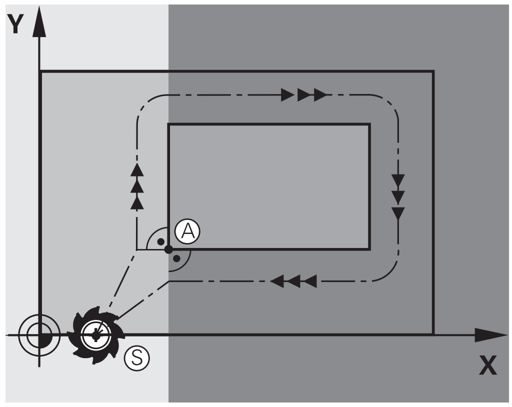
5.3 Approaching and departing a contour

Starting point and end point

The tool approaches the first contour point from the starting point. The starting point must be:

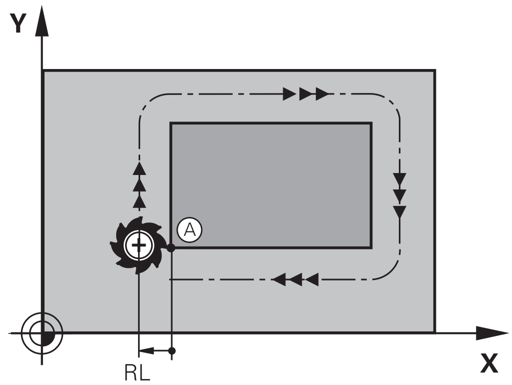
- Programmed without radius compensation
- Approachable without danger of collision
- Close to the first contour point

Example in the figure on the right:
If you set the starting point in the dark gray area, the contour will be damaged when the first contour element is approached.



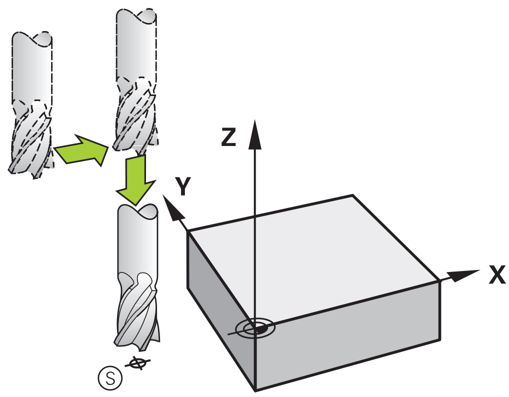
First contour point

You need to program a radius compensation for the tool movement to the first contour point.



Approaching the starting point in the spindle axis

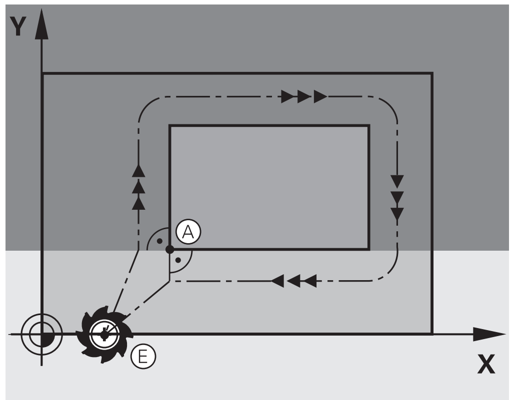
When the starting point is approached, the tool must be moved to the working depth in the spindle axis. If danger of collision exists, approach the starting point in the spindle axis separately.



Example

```
30 L  Z-10  R0  FMAX
31 L X+20  Y+30  RL  F350
```

End point

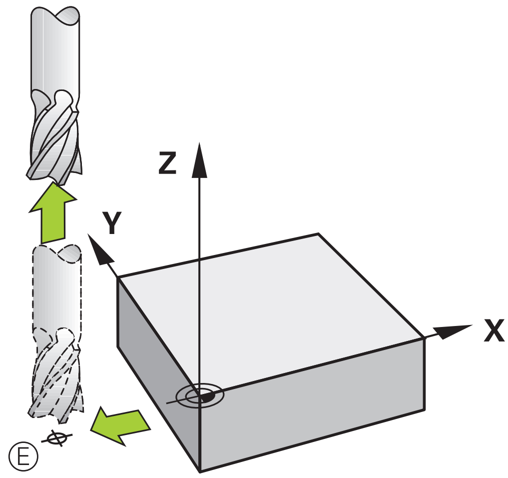


The end point should be selected so that it is:

- Approachable without danger of collision
- Near to the last contour point
- In order to make sure the contour will not be damaged, the optimal ending point should lie on the extended tool path for machining the last contour element

Example in the figure on the right:
If you set the end point in the dark gray area, the contour will be damaged when the end point is approached.

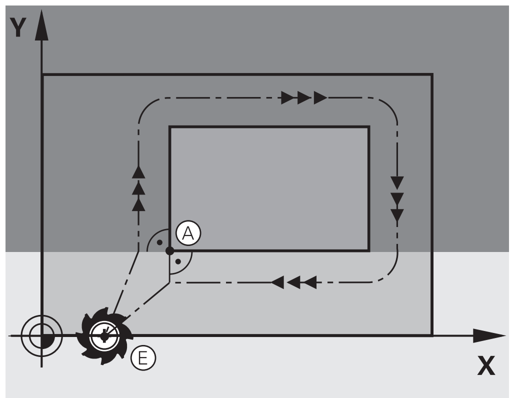
Departing the end point in the spindle axis:
Program the departure from the end point in the spindle axis separately.



Example

50 L X+60 Y+70 R0 F700	
51 L Z+250 R0 FMAX	

Common starting and end points



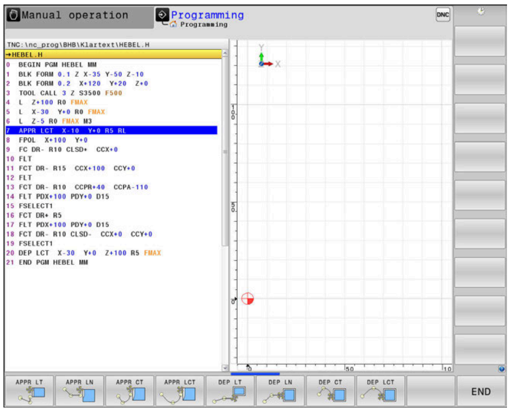
Do not program any radius compensation if the starting point and end point are the same.

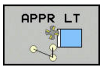
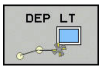
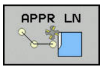
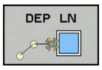
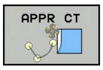
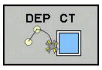
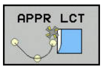
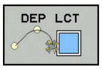
In order to make sure the contour will not be damaged, the optimal starting point should lie between the extended tool paths for machining the first and last contour elements.

Example in the figure on the right:
If you set the end point in the dark gray area, the contour will be damaged when the contour is approached/departed.

Overview: Types of paths for contour approach and departure

The functions for contour approach **APPR** and departure **DEP** are activated with the **APPR/DEP** key. You can then select the following path forms with the corresponding soft keys:

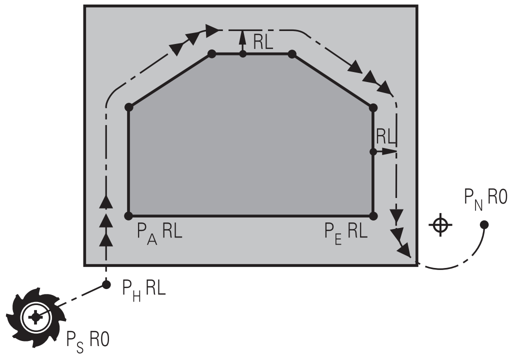


Approach	Departure	Function
		Straight line with tangential connection
		Straight line perpendicular to a contour point
		Circular arc with tangential connection
		Circular arc with tangential connection to the contour. Approach and departure to an auxiliary point outside the contour on a tangentially connecting line

Approaching and departing a helix

The tool approaches and departs a helix on its extension by moving in a circular arc that connects tangentially to the contour. You program helical approach and departure with the **APPR CT** and **DEP CT** functions.

Important positions for approach and departure



NOTICE

Danger of collision!

The control traverses from the current position (starting point P_S) to the auxiliary point P_H at the last feed rate entered. If you programmed **FMAX** in the last positioning block before the approach function, the control also approaches the auxiliary point P_H at rapid traverse.

- Program a feed rate other than **FMAX** before the approach function

- Starting point P_S
  You program this position in the block before the APPR block. P_S lies outside the contour and is approached without radius compensation (R0).
- Auxiliary point P_H
  Some of the paths for approach and departure go through an auxiliary point P_H that the control calculates from your input in the APPR or DEP block.
- First contour point P_A and last contour point P_E
  You program the first contour point P_A in the APPR block. The last contour point P_E can be programmed with any path function. If the APPR block also includes the Z coordinate, the control moves the tool simultaneously to the first contour point P_A.
- End point P_N
  The position P_N lies outside of the contour and results from your input in the DEP block. If the DEP block also includes the Z coordinate, the control moves the tool simultaneously to the end point P_N.

Designation	Meaning
APPR	Approach
DEP	Departure
L	Line
C	Circle
T	Tangential (smooth connection)
N	Normal (perpendicular)

NOTICE

Danger of collision!

The control does not automatically check whether collisions can occur between the tool and the workpiece. Incorrect pre-positioning and incorrect auxiliary points P_H can also lead to contour damage. There is danger of collision during the approach movement!

- Program a suitable pre-position
- Check the auxiliary point P_H, the sequence and the contour with the aid of the graphic simulation

With the **APPR LT**, **APPR LN** and **APPR CT** functions, the control moves the tool to the auxiliary point P_H at the last programmed feed rate (which can also be **FMAX**). With the **APPR LCT** function, the control moves to the auxiliary point P_H at the feed rate programmed with the APPR block. If no feed rate is programmed yet before the approach block, the control generates an error message.

Polar coordinates

You can also program the contour points for the following approach/ departure functions over polar coordinates:

- APPR LT becomes APPR PLT
- APPR LN becomes APPR PLN
- APPR CT becomes APPR PCT
- APPR LCT becomes APPR PLCT
- DEP LCT becomes DEP PLCT

Select by soft key an approach or departure function, then press the orange **P** key.

Radius compensation

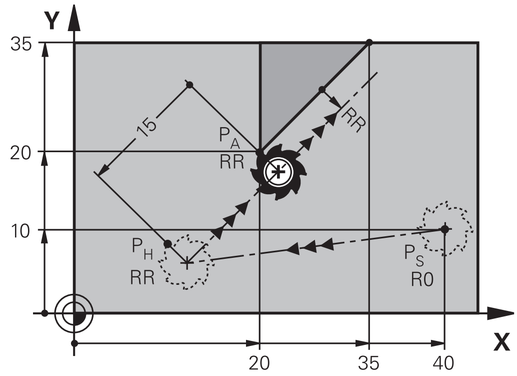
The tool radius compensation is programmed together with the first contour point P_A in the APPR block. The DEP blocks automatically discard the tool radius compensation.

If you program **APPR LN** or **APPR CT** with **R0**, the control stops the machining/simulation with an error message.

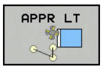
This method of function differs from the iTNC 530 control!

Approaching on a straight line with tangential connection: APPR LT

The tool moves on a straight line from the starting point P_S to an auxiliary point P_H. It then moves to the first contour point P_A on a straight line that connects tangentially to the contour. The auxiliary point P_H is separated from the first contour point P_A by the distance **LEN**.



- Use any path function to approach the starting point P_S
- Initiate the dialog with the **APPR DEP** key and **APPR LT** soft key



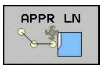
  - Coordinates of the first contour point P_A
  - **LEN**: Distance from the auxiliary point P_H to the first contour point P_A
  - Radius compensation **RR/RL** for machining

Example

7 L X+40 Y+10 R0 FMAX M3	P_S without radius compensation
8 APPR LT X+20 Y+20 Z-10 LEN15 RR F100	P_A with radius comp. RR, distance P_H to P_A: LEN=15
9 L X+35 Y+35	End point of the first contour element
10 L ...	Next contour element

Approaching on a straight line perpendicular to the first contour point: APPR LN

- Use any path function to approach the starting point P_S.
- Initiate the dialog with the **APPR DEP** key and **APPR LN** soft key:

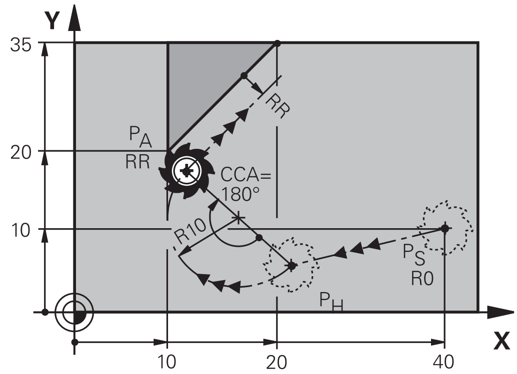


  - Coordinates of the first contour point P_A
  - Length: Distance to the auxiliary point P_H. Always enter **LEN** as a positive value
  - Radius compensation **RR/RL** for machining

Example

7 L X+40 Y+10 R0 FMAX M3	Approach PS without radius compensation
8 APPR LN X+10 Y+20 Z-10 LEN15 RR F100	PA with radius comp. RR
9 L X+20 Y+35	End point of the first contour element
10 L ...	Next contour element

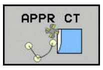
Approaching on a circular path with tangential connection: APPR CT



The tool moves on a straight line from the starting point P_S to an auxiliary point P_H. It then moves from PH to the first contour point PA following a circular arc that is tangential to the first contour element.

The arc from P_H to P_A is determined through the radius R and the center angle **CCA**. The direction of rotation of the circular arc is automatically derived from the tool path for the first contour element.

- Use any path function to approach the starting point P_S.
- Initiate the dialog with the **APPR DEP** key and **APPR CT** soft key

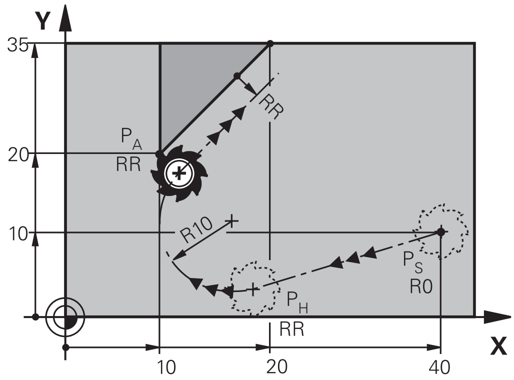


- Coordinates of the first contour point P_A
- Radius R of the circular arc
  - If the tool should approach the workpiece in the direction defined by the radius compensation: Enter R as a positive value
  - If the tool should approach the workpiece opposite to the radius compensation: Enter R as a negative value.
- Center angle **CCA** of the arc
  - CCA can be entered only as a positive value.
  - Maximum input value 360°
- Radius compensation **RR/RL** for machining

Example

7 L X+40 Y+10 R0 FMAX M3	Approach PS without radius compensation
8 APPR CT X+10 Y+20 Z-10 CCA180 R+10 RR F100	PA with radius compensation RR, radius R=10
9 L X+20 Y+35	End point of the first contour element
10 L ...	Next contour element

Approaching on a circular path with tangential connection from a straight line to the contour: APPR LCT

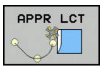


The tool moves on a straight line from the starting point P_S to an auxiliary point P_H. It then moves to the first contour point P_A on a circular arc. The feed rate programmed in the APPR block is effective for the entire path that the control traversed in the approach block (path P_S to P_A).

If you have programmed the coordinates of all three principal axes X, Y and Z in the approach block, the control moves the tool from the position defined before the APPR block to the auxiliary point P_H on all three axes simultaneously. Then the connect goes from P_H to P_A only on the working plane.

The arc is connected tangentially both to the line P_S - P_H as well as to the first contour element. Once these lines are known, the radius then suffices to completely define the tool path.

- Use any path function to approach the starting point P_S.
- Initiate the dialog with the **APPR DEP** key and **APPR LCT** soft key:



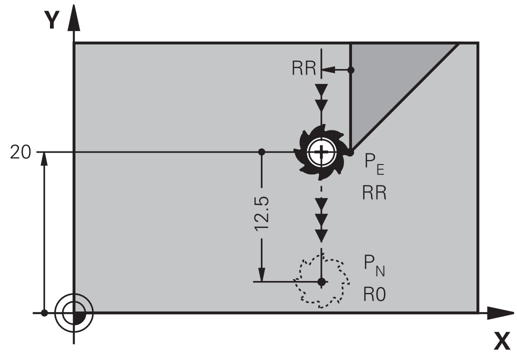
- Coordinates of the first contour point P_A
- Radius R of the circular arc. Enter R as a positive value
- Radius compensation **RR/RL** for machining

Example

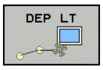
7 L X+40 Y+10 R0 FMAX M3	Approach PS without radius compensation
8 APPR LCT X+10 Y+20 Z-10 R10 RR F100	PA with radius compensation RR, radius R=10
9 L X+20 Y+35	End point of the first contour element
10 L ...	Next contour element

Departing in a straight line with tangential connection: DEP LT

The tool moves on a straight line from the last contour point P_E to the end point P_N. The line lies on the extension of the last contour element. P_N is separated from P_E by the distance **LEN**.



- Program the last contour element with the end point P_E and radius compensation
- Initiate the dialog with the **APPR DEP** key and **DEP LT** soft key



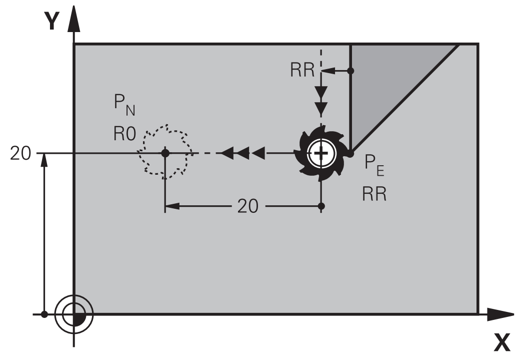
- LEN: Enter the distance from the last contour element P_E to the end point P_N.

Example

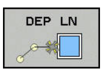
23 L Y+20 RR F100	Last contour element: PE with radius compensation
24 DEP LT LEN12.5 F100	Depart contour by LEN=12.5 mm
25 L Z+100 FMAX M2	Retract in Z, return jump, end program

Departing in a straight line perpendicular to the last contour point: DEP LN

The tool moves on a straight line from the last contour point P_E to the end point P_N. The line departs on a perpendicular path from the last contour point P_E. P_N is separated from P_E by the distance **LEN** plus the tool radius.



- Program the last contour element with the end point P_E and radius compensation
- Initiate the dialog with the **APPR DEP** key and **DEP LN** soft key



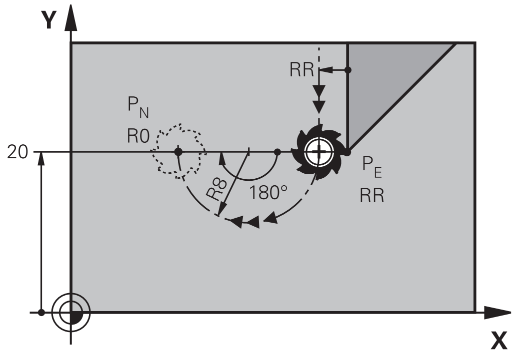
- **LEN**: Enter the distance from the last contour element to P_N. Important: Enter a positive value in **LEN**

Example

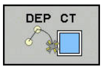
23 L Y+20 RR F100	Last contour element: PE with radius compensation
24 DEP LN LEN+20 F100	Depart perpendicular to contour by LEN=20 mm
25 L Z+100 FMAX M2	Retract in Z, return jump, end program

Departing on a circular path with tangential connection: DEP CT

The tool moves on a circular arc from the last contour point P_E to the end point P_N. The circular arc connects tangentially to the last contour element.



- Program the last contour element with the end point P_E and radius compensation
- Initiate the dialog with the **APPR DEP** key and **DEP CT** soft key



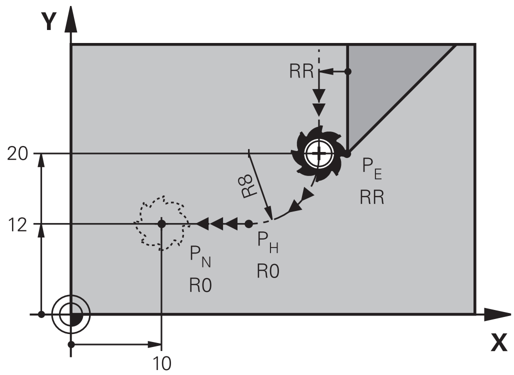
- Center angle **CCA** of the arc
- Radius R of the circular arc
  - If the tool should depart the workpiece in the direction opposite to the radius compensation: Enter R as a positive value.
  - If the tool should depart the workpiece in the direction **opposite** to the radius compensation: Enter R as a negative value.

Example

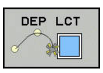
23 L Y+20 RR F100	Last contour element: PE with radius compensation
24 DEP CT CCA 180 R+8 F100	Center angle=180°, arc radius=8 mm
25 L Z+100 FMAX M2	Retract in Z, return jump, end program

Departing on a circular arc tangentially connecting the contour and a straight line: DEP LCT

The tool moves on a circular arc from the last contour point P_S to an auxiliary point P_H. It then moves on a straight line to the end point P_N. The arc is tangentially connected both to the last contour element and to the line from P_H to P_N. Once these lines are known, the radius R suffices to unambiguously define the tool path.



- Program the last contour element with the end point P_E and radius compensation
- Initiate the dialog with the **APPR/DEP** key and **DEP LCT** soft key



- Enter the coordinates of the end point P_N
- Radius R of the circular arc. Enter R as a positive value

Example

23 L Y+20 RR F100	Last contour element: PE with radius compensation
24 DEP LCT X+10 Y+12 R+8 F100	Coordinates PN, arc radius=8 mm
25 L Z+100 FMAX M2	Retract in Z, return jump, end program

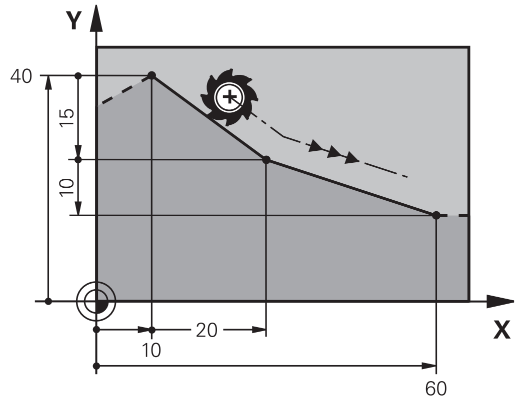
5.4 Path contours — Cartesian coordinates

Overview of path functions

Key	Function	Tool movement	Required input	Page
L	Straight line **L**	Straight line	Coordinates of the end point	155
CHF	Chamfer: **CHF**	Chamfer between two straight lines	Chamfer side length	156
CC	Circle center **CC**	None	Coordinates of the circle center or pole	158
C	Circular arc **C**	Circular arc around a circle center CC to an arc end point	Coordinates of the arc end point, direction of rotation	159
CR	Circular arc **CR**	Circular arc with a certain radius	Coordinates of the arc end point, arc radius, direction of rotation	160
CT	Circular arc **CT**	Circular arc with tangential connection to the preceding and subsequent contour elements	Coordinates of the arc end point	162
RND	Corner rounding **RND**	Circular arc with tangential connection to the preceding and subsequent contour elements	Rounding radius R	157
FK	**FK** free contour programming	Straight line or circular path with any connection to the preceding contour element	Input depends on the function	176

Straight line L

The control moves the tool in a straight line from its current position to the straight-line end point. The starting point is the end point of the preceding NC block.



- Press the **L** key to open a program block for a linear movement
- **Coordinates** of the end point of the straight line, if necessary
- **Radius compensation RL/RR/R0**
- **Feed rate F**
- **Miscellaneous function M**

Example

7 L X+10 Y+40 RL F200 M3
8 L IX+20 IY-15
9 L X+60 IY-10

Actual position capture

You can also generate a straight-line block (**L** block) by using the **actual position capture** key:

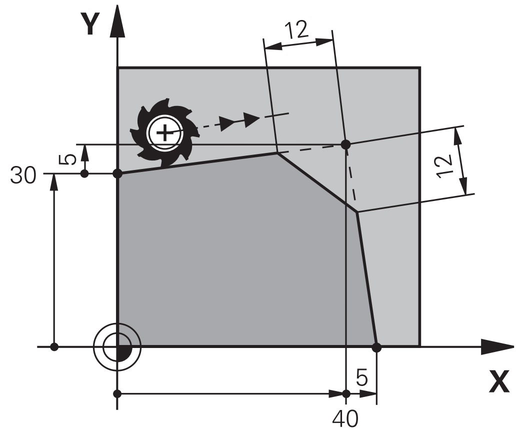
- In **Manual Operation** mode, move the tool to the position you want to capture
- Switch the screen display to programming
- Select the NC block after which you want to insert the straight line block

- Press the **actual position capture** key
- > The control generates a straight-line block with the actual position coordinates.

Inserting a chamfer between two straight lines

The chamfer enables you to cut off corners at the intersection of two straight lines.

- The line blocks before and after the **CHF** block must be in the same working plane as the chamfer.
- The radius compensation before and after the **CHF** block must be the same
- The chamfer must be machinable with the current tool



CHF

- **Chamfer side length**: Length of the chamfer, and if necessary:
- **Feed rate F** (effective only in **CHF** block)

Example

7 L X+0 Y+30 RL F300 M3
8 L X+40 IY+5
9 CHF 12 F250
10 L IX+5 Y+0

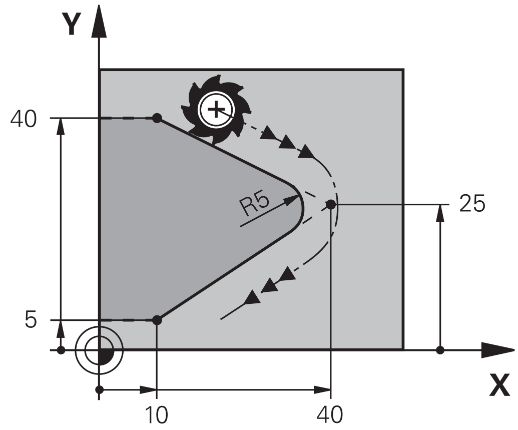
You cannot start a contour with a **CHF** block.
A chamfer is possible only in the working plane.
The corner point is cut off by the chamfer and is not part of the contour.
A feed rate programmed in the **CHF** block is effective only in that CHF block. After the **CHF** block, the previous feed rate becomes effective again.

Rounded corners RND

The **RND** function creates rounding arcs at contour corners.

The tool moves on an arc that connects tangentially to both the preceding and subsequent contour elements.

The rounding arc must be machinable with the called tool.



- **Rounding radius**: Enter the radius, and if necessary:
- **Feed F** (effective only in the **RND** block)

Example

5 L X+10 Y+40 RL F300 M3
6 L X+40 Y+25
7 RND R5 F100
8 L X+10 Y+5

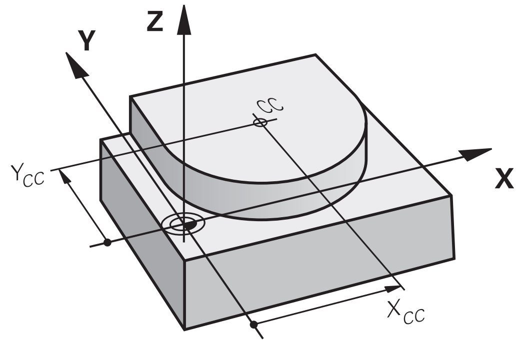
In the preceding and subsequent contour elements, both coordinates must lie in the plane of the rounding arc. If you machine the contour without tool-radius compensation, you must program both coordinates in the plane.

The tool will not move to the corner point.

A feed rate programmed in the **RND** block is effective only in that **RND** block. After the **RND** block, the previous feed rate becomes effective again.

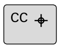
You can also use an **RND** block for a tangential contour approach.

Circle center CC



You can define a circle center for circles that you have programmed with the C key (circular path C) This is done in the following ways:

- Enter the Cartesian coordinates of the circle center in the working plane, or
- Use the position last programmed, or
- Take over the coordinates with the **Actual-position-capture** key



- Enter coordinates for the circle center or, if you want to use the last programmed position, enter no coordinates

Example

```
5 CC  X+25 Y+25
```

or

```
10 L X+25 Y+25
11 CC
```

The program lines 10 and 11 do not refer to the illustration.

Validity

The circle center definition remains in effect until you program a new circle center.

Entering the circle center incrementally

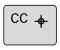
If you enter the circle center with incremental coordinates, you have programmed it relative to the last programmed position of the tool.

> The only effect of **CC** is to define a position as circle center: The tool does not move to this position.
>
> The circle center is also the pole for polar coordinates.

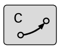
Circular arc C around circle center CC

Before programming a circular arc C, you must first specify the circle center **CC**. The last programmed tool position will be the starting point of the arc.

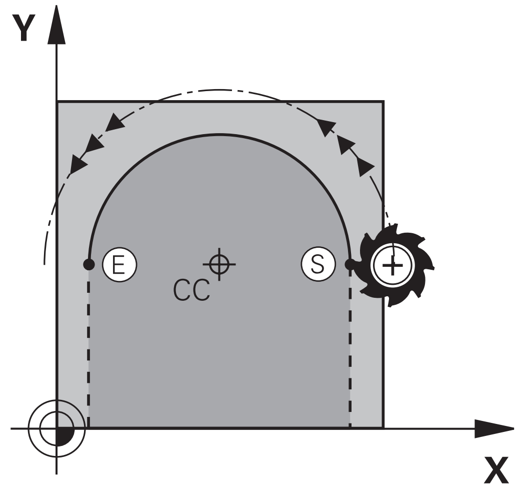
- Move the tool to the starting point of the circle



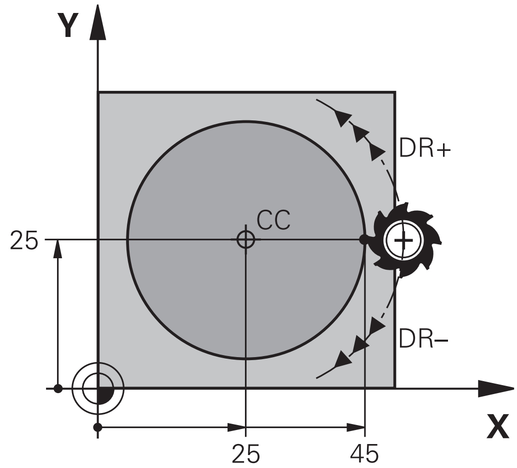
- Enter the **coordinates** of the circle center



- Enter the **coordinates** of the arc end point, if necessary:
- **Direction of rotation DR**
- **Feed F**
- **Miscellaneous function M**



> The control normally makes circular movements in the active working plane. However, you can also program circular arcs that do not lie in the active working plane. By simultaneously rotating these circular movements you can create spatial arcs (arcs in three axes), e.g. **C Z... X... DR+** (with tool axis Z).



Example

```
5 CC  X+25 Y+25
6 L X+45 Y+25 RR F200 M3
7 C X+45 Y+25 DR+
```

Full circle

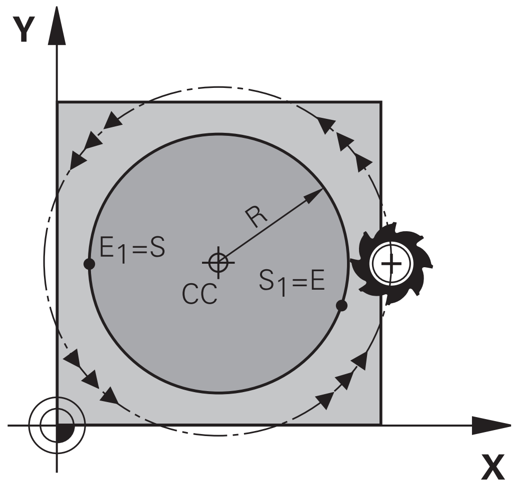
For the end point, program the same coordinates as for the starting point.

> The starting and end points of the arc must lie on the circle.
>
> The maximum value for input tolerance is 0.016 mm. Set the input tolerance in the machine parameter **circleDeviation** (no. 200901).
>
> Smallest possible circle that the control can traverse: 0.016 mm.

Circular arc CR with fixed radius

The tool moves on a circular path with the radius R.

- **Coordinates** of the arc end point
- **Radius R** Caution: The algebraic sign determines the size of the arc!
- **Direction of rotation DR** Caution: The algebraic sign determines whether the arc is concave or convex.
- **Miscellaneous function M**
- **Feed F**



Full circle

For a full circle, program two semicircle blocks in succession:

The end point of the first semicircle is the starting point of the second. The end point of the second semicircle is the starting point of the first.

Central angle CCA and arc radius R

The starting and end points on the contour can be connected with four arcs of the same radius:

Smaller arc: CCA<180°

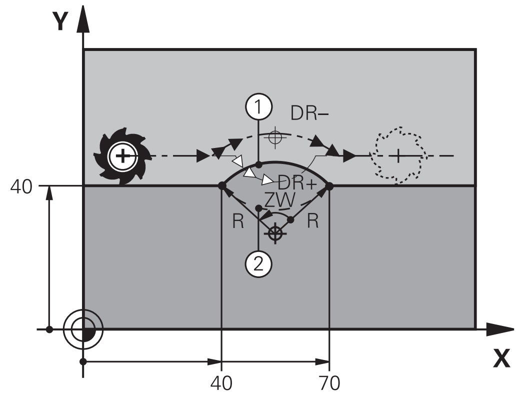
Enter the radius with a positive sign, i.e. R>0

Larger arc: CCA>180°
Enter the radius with a negative sign, i.e. R<0

The direction of rotation determines whether the arc is curving outward (convex) or curving inward (concave):

Convex: Direction of rotation **DR-** (with radius compensation **RL**)

Concave: Direction of rotation **DR+** (with radius compensation **RL**)



> The distance from the starting and end points of the arc diameter cannot be greater than the diameter of the arc.
>
> The maximum radius is 99.9999 m.
>
> You can also enter rotary axes A, B and C.
>
> The control normally makes circular movements in the active working plane. However, you can also program circular arcs that do not lie in the active working plane. By simultaneously rotating these circular movements you can create spatial arcs (arcs in three axes).

Example

```
10 L X+40 Y+40 RL F200 M3
11 CR  X+70 Y+40  R+20 DR-  (arc 1)
```

or

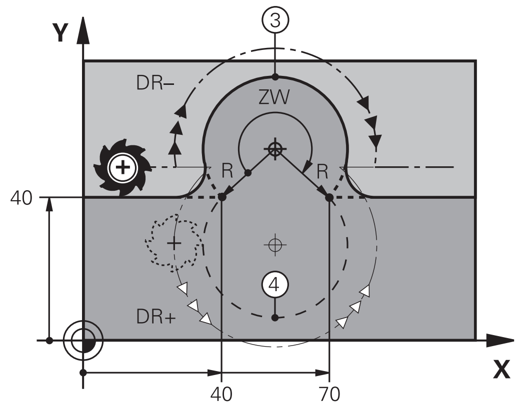
```
11 CR  X+70 Y+40  R+20 DR+  (arc 2)
```

or

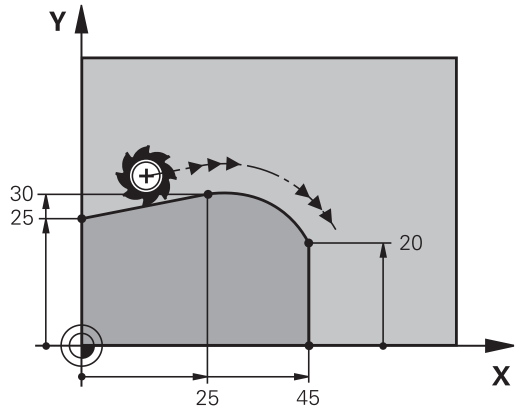
```
11 CR X+70 Y+40 R-20 DR- (arc 3)
```

or

```
11 CR X+70 Y+40 R-20 DR+ (arc 4)
```



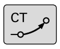
Circular arc CT with tangential transition



The tool moves on an arc that connects tangentially to the previously programmed contour element.

A connection between two contour elements is called tangential when there is no kink or corner at the intersection between the two contours—the transition is smooth.

The contour element to which the tangential arc connects must be programmed immediately before the **CT** block. This requires at least two positioning blocks.



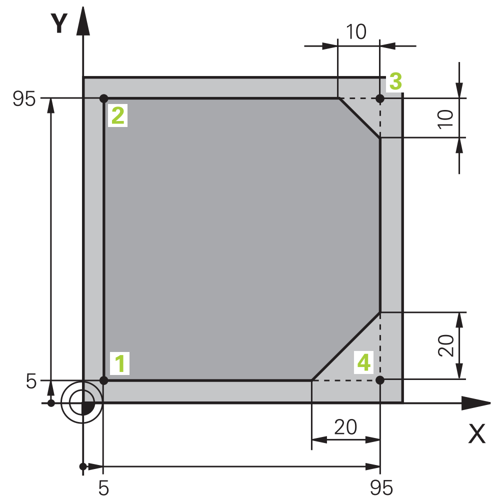
- **Coordinates** of the arc end point, and if necessary:
- **Feed F**
- **Miscellaneous function M**

Example

7 L X+0 Y+25 RL F300 M3
8 L X+25 Y+30
9 CT X+45 Y+20
10 L Y+0

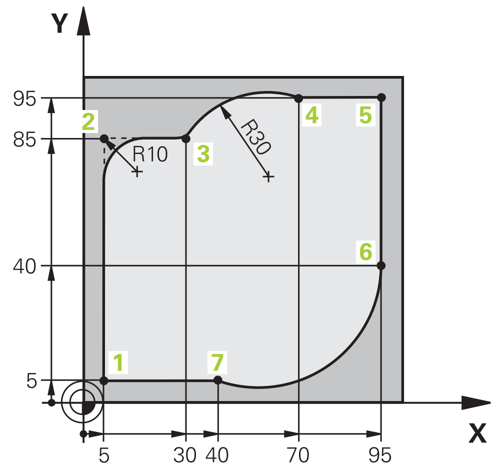
> A tangential arc is a two-dimensional operation: the coordinates in the **CT** block and in the contour element preceding it must be in the same plane of the arc!

Example: Linear movements and chamfers with Cartesian coordinates



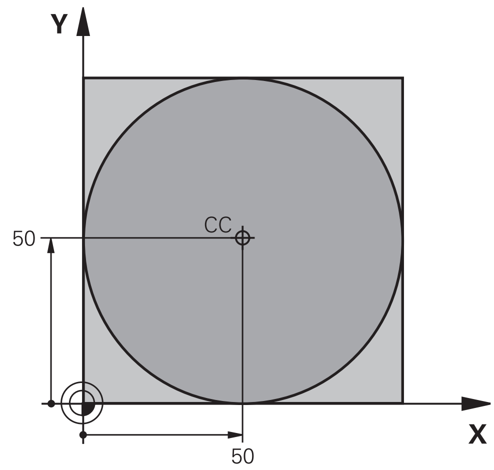
0 BEGIN PGM LINEAR MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	Define the workpiece blank for graphic workpiece simulation
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 1 Z S4000	Call the tool in the spindle axis and with spindle speed
4 L Z+250 R0 FMAX	Retract the tool in the spindle axis at rapid traverse FMAX
5 L X-10 Y-10 R0 FMAX	Pre-position the tool
6 L Z-5 R0 F1000 M3	Move to working depth at feed rate F = 1000 mm/min
7 APPR LT X+5 y+5 LEN10 RL F300	Approach the contour at point 1 on a straight line with tangential connection
8 L Y+95	Move to point 2
9 L X+95	Point 3: first straight line for corner 3
10 CHF 10	Program a chamfer with length 10 mm
11 L Y+5	Point 4: 2nd straight line for corner 3, 1st straight line for corner 4
12 CHF 20	Program a chamfer with length 20 mm
13 L X+5	Move to last contour point 1, second straight line for corner 4
14 DEP LT LEN10 F1000	Depart the contour on a straight line with tangential connection
15 L Z+250 R0 FMAX M2	Retract the tool, end program
16 END PGM LINEAR MM	

Example: Circular movements with Cartesian coordinates



0 BEGIN PGM CIRCULAR MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	Define the workpiece blank for graphic workpiece simulation
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 1 Z s4000	Call the tool in the spindle axis and with spindle speed
4 L Z+250 R0 FMAX	Retract the tool in the spindle axis at rapid traverse FMAX
5 L X-10 Y-10 R0 FMAX	Pre-position the tool
6 L Z-5 R0 F1000 M3	Move to working depth at feed rate F = 1000 mm/min
7 APPR LCT X+5 Y+5 R5 RL F300	Approach the contour at point 1 on a circular arc with tangential connection
8 L X+5 Y+85	Point 2: First straight line for corner 2
9 RND R10 F150	Insert radius with R = 10 mm, feed rate: 150 mm/min
10 L X+30 Y+85	Move to point 3: Starting point of the arc with CR
11 CR X+70 Y+95 R+30 DR-	Move to point 4: End point of the arc with CR, radius 30 mm
12 L X+95	Move to point 5
13 L X+95 Y+40	Move to point 6
14 CT X+40 Y+5	Move to point 7: End point of the arc, circular arc with tangential connection to point 6, the control automatically calculates the radius
15 L X+5	Move to last contour point 1
16 DEP LCT X-20 Y-20 R5 F1000	Depart the contour on a circular arc with tangential connection
17 L Z+250 R0 FMAX M2	Retract the tool, end program
18 END PGM CIRCULAR MM	

Example: Full circle with Cartesian coordinates



0 BEGIN PGM C-CC MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	Workpiece blank definition
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 1 Z S3150	Tool call
4 CC X+50 Y+50	Define the circle center
5 L Z+250 R0 FMAX	Retract the tool
6 L X-40 Y+50 R0 FMAX	Pre-position the tool
7 L Z-5 R0 F1000 M3	Move to working depth
8 APPR LCT X+0 Y+50 R5 RL F300	Approach the starting point of the circle on a circular arc with tangential connection
9 C X+0 DR-	Move to the circle end point (= circle starting point)
10 DEP LCT X-40 Y+50 R5 F1000	Depart the contour on a circular arc with tangential connection
11 L Z+250 R0 FMAX M2	Retract the tool, end program
12 END PGM C-CC MM	

5.5 Path contours – Polar coordinates

Overview

With polar coordinates you can define a position in terms of its angle **PA** and its distance **PR** relative to a previously defined pole **CC**.

Polar coordinates are useful with:

- Positions on circular arcs
- Workpiece drawing dimensions in degrees, e.g. bolt hole circles

Overview of path functions with polar coordinates

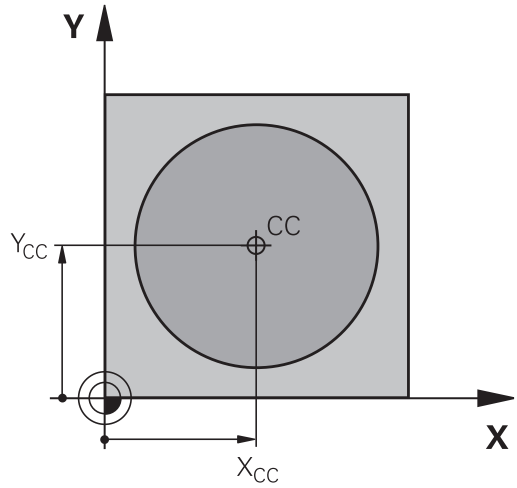
Key	Tool movement	Required input	Page
L + P	Straight line	Polar radius, polar angle of the straight-line end point	167
C + P	Circular path around circle center/pole to arc end point	Polar angle of the arc end point, direction of rotation	168
CT + P	Circular arc with tangential connection to the preceding contour element	Polar radius, polar angle of the arc end point	168
C + P	Combination of a circular and a linear movement	Polar radius, polar angle of the arc end point, coordinate of the end point in the tool axis	169

Datum for polar coordinates: pole CC

You can set the pole CC at any point in the NC program, before indicating positions in polar coordinates. Set the pole in the same way as you would program the circle center.

CC

- **Coordinates**: Enter Cartesian coordinates for the pole or, if you want to use the last programmed position, do not enter any coordinates. Before programming polar coordinates, define the pole. You can only define the pole in Cartesian coordinates. The pole remains in effect until you define a new pole.



Example

12 CC X+45 Y+25

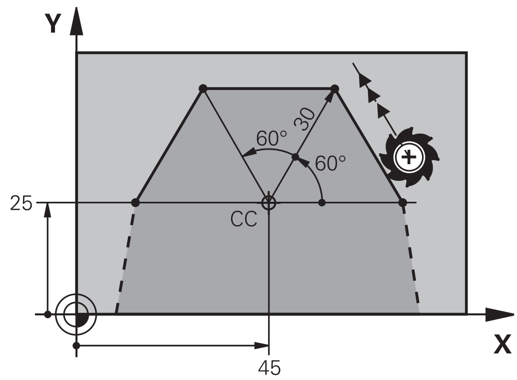
Straight line LP

The tool moves in a straight line from its current position to the straight-line end point. The starting point is the end point of the preceding NC block.

L

P

- **Polar coordinate radius PR**: Enter the distance from the pole CC to the straight-line end point
- **Polar-coordinates angle PA**: Angular position of the straight-line end point between –360° and +360°



The sign of **PA** depends on the angle reference axis:

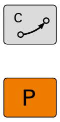
- If the angle from the angle reference axis to **PR** is counterclockwise: **PA**>0
- If the angle from the angle reference axis to **PR** is clockwise: **PA**<0

Example

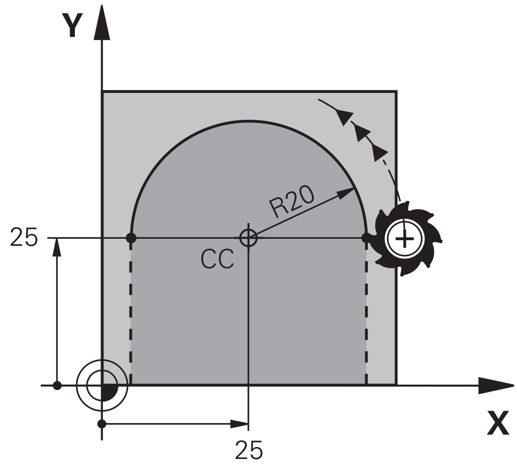
12 CC X+45 Y+25
13 LP PR+30 PA+0 RR F300 M3
14 LP PA+60
15 LP IPA+60
16 LP PA+180

Circular path CP around pole CC

The polar coordinate radius **PR** is also the radius of the arc. **PR** is defined by the distance from the starting point to the pole **CC**. The last programmed tool position will be the starting point of the arc.



- **Polar-coordinates angle PA**: Angular position of the arc end point between –99999.9999° and +99999.9999°
- **Direction of rotation DR**



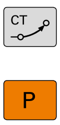
Example

18 CC X+25 Y+25
19 LP PR+20 PA+0 RR F250 M3
20 CP PA+180 DR+

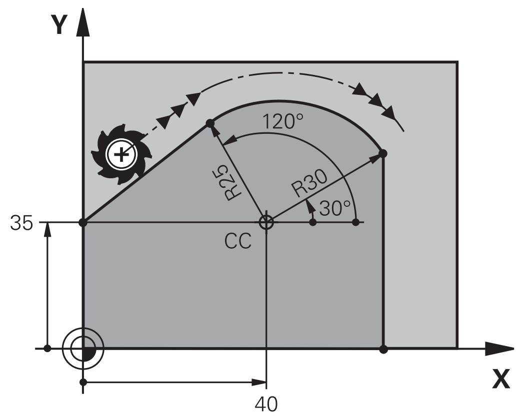
> With incremental inputs you must enter DR and PA with the same sign.
> Consider this behavior when importing NC programs from earlier controls. Adapt the NC program if required.

Circle CTP with tangential connection

The tool moves on a circular path, starting tangentially from a preceding contour element.



- **Polar coordinate radius PR**: Distance between the arc end point and the pole **CC**
- **Polar coordinate angle PA**: Angular position of the arc end point.



> The pole is **not** the center of the contour arc!

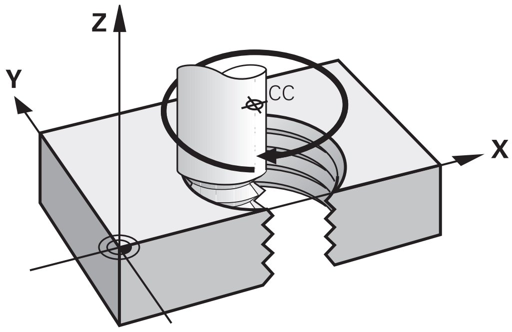
Example

12 CC X+40 Y+35
13 L X+0 Y+35 RL F250 M3
14 LP PR+25 PA+120
15 CTP PR+30 PA+30
16 L Y+0

Helix

A helix is a combination of a circular movement in a main plane and a linear movement perpendicular to this plane. You program the circular path in a main plane.

A helix is programmed only in polar coordinates.



Application

- Large-diameter internal and external threads
- Lubrication grooves

Calculating the helix

To program a helix, you must enter the total angle through which the tool is to move on the helix in incremental dimensions, and the total height of the helix.

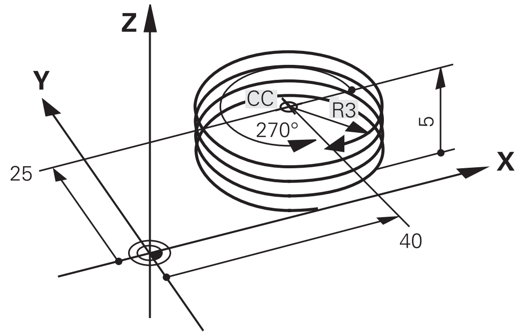
Thread revolutions n:	Thread revolutions + thread overrun at the start and end of the thread
Total height h:	Thread pitch P times thread revolutions n
Incremental total angle **IPA**:	Thread revolutions x 360° + angle for beginning of thread + angle for thread overrun
Starting coordinate Z:	Pitch P times (thread revolutions + thread overrun at start of thread)

Shape of the helix

The table below illustrates in which way the shape of the helix is determined by the work direction, direction of rotation and radius compensation.

Internal thread	Work direction	Direction of rotation	Radius compensation
Right-hand Left-hand	Z+ Z+	**DR+** **DR-**	**RL** **RR**
Right-hand Left-hand	Z– Z–	**DR-** **DR+**	**RR** **RL**
External thread			
Right-hand Left-hand	Z+ Z+	**DR+** **DR-**	**RR** **RL**
Right-hand Left-hand	Z– Z–	**DR-** **DR+**	**RL** **RR**

Programming a helix



> Always enter the same algebraic sign for the direction of rotation and the incremental total angle **IPA**. The tool may otherwise move in a wrong path and damage the contour.
>
> For the total angle **IPA** you can enter a value of -99 999.9999° to +99 999.9999°.

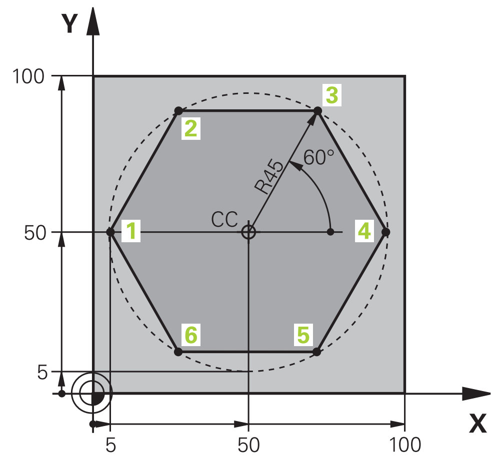
P

- **Polar coordinates angle:** Enter the total angle of tool traverse along the helix in incremental dimensions.
- **After entering the angle, specify the tool axis with an axis selection key**
- **Coordinate**: Enter the coordinate for the height of the helix in incremental dimensions
- **Direction of rotation DR**
  Clockwise helix: DR–
  Counterclockwise helix: DR+
- **Enter the radius compensation** according to the table

Example: Thread M6 x 1 mm with 5 revolutions

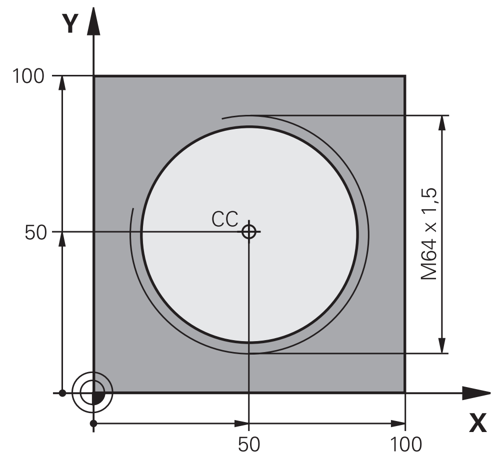
```
12 CC X+40 Y+25
13 L  Z+0 F100  M3
14 LP PR+3 PA+270 RL F50
15 CP  IPA-1800 IZ+5  DR-
```

Example: Linear movement with polar coordinates



0 BEGIN PGM LINEARPO MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	Workpiece blank definition
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 1 Z S4000	Tool call
4 CC X+50 Y+50	Define the preset for polar coordinates
5 L Z+250 R0 FMAX	Retract the tool
6 LP PR+60 PA+180 R0 FMAX	Pre-position the tool
7 L Z-5 R0 F1000 M3	Move to working depth
8 APPR PLCT PR+45 PA+180 R5 RL F250	Approach the contour at point 1 on a circular arc with tangential connection
9 LP PA+120	Move to point 2
10 LP PA+60	Move to point 3
11 LP PA+0	Move to point 4
12 LP PA-60	Move to point 5
13 LP PA-120	Move to point 6
14 LP PA+180	Move to point 1
15 DEP PLCT PR+60 PA+180 R5 F1000	Depart the contour on a circular arc with tangential connection
16 L Z+250 R0 FMAX M2	Retract the tool, end of program
17 END PGM LINEARPO MM	

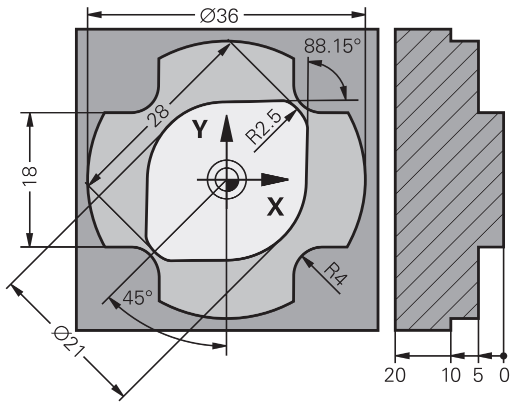
Example: Helix



0 BEGIN PGM HELIX MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	Workpiece blank definition
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 1 Z S1400	Tool call
4 L Z+250 R0 FMAX	Retract the tool
5 L X+50 Y+50 R0 FMAX	Pre-position the tool
6 CC	Transfer the last programmed position as the pole
7 L Z-12.75 R0 F1000 M3	Move to working depth
8 APPR PCT PR+32 PA-182 CCA180 R+2 RL F100	Approach the contour on a circular arc with tangential connection
9 CP IPA+3240 IZ+13.5 DR+ F200	Helical interpolation
10 DEP CT CCA180 R+2	Depart the contour on a circular arc with tangential connection
11 L Z+250 R0 FMAX M2	Retract the tool, end of program
12 END PGM HELIX MM	

5.6 Path contours – FK free contour programming

Fundamentals



Workpiece drawings that are not dimensioned for NC often contain unconventional coordinate data that cannot be entered with the gray path function keys.

You can enter such dimensional data directly by using the free contour programming function FK, e.g.

- If there are known coordinates on or in proximity to the contour element
- If coordinate data refers to another contour element
- If directional data and data regarding the course of the contour are known

The control derives the contour from the known coordinate data and supports the programming dialog with the interactive FK programming graphics. The figure at upper right shows a workpiece drawing for which FK programming is the most convenient programming method.

Programming notes

You must enter all available data for every contour element. Even the data that does not change must be entered in every NC block—otherwise it will not be recognized.

Q parameters are permissible in all FK elements, except in elements with relative references (e.g. **RX** or **RAN**), or in elements that are referenced to other NC blocks.

If both FK blocks and conventional blocks are entered in an NC program, the FK contour must be fully defined before you can return to conventional programming.

Program all of the contours before combining them (e.g., with the SL cycles). You thereby ensure that the contours are correctly defined and avoid unnecessary error messages.

The control needs a fixed point that it can use as the basis for all calculations. Use the gray path function keys to program a position that contains both coordinates of the working plane immediately before programming the FK contour. Do not enter any Q parameters in this NC block.

If the first NC block of an FK contour is an **FCT** or **FLT** block, you must program at least two NC blocks with the gray path function keys before it. This fully defines the approach direction.

Do not program an FK contour immediately after an **LBL** command.

You cannot combine the cycle call **M89** with FK programming.

Defining the working plane

The FK free contour programming feature can only be used for programming contour elements that lie in the working plane.

The control defines the working plane for FK programming according to the following hierarchy:

1 Through the plane defined in a **FPOL** block
2 In the Z/X plane if the FK sequence is performed in turning mode
3 Through the working plane specified and defined in the **TOOL CALL** (e.g., **TOOL CALL 1 Z** = X/Y plane)
4 If none of this applies, then the standard X/Y plane is active

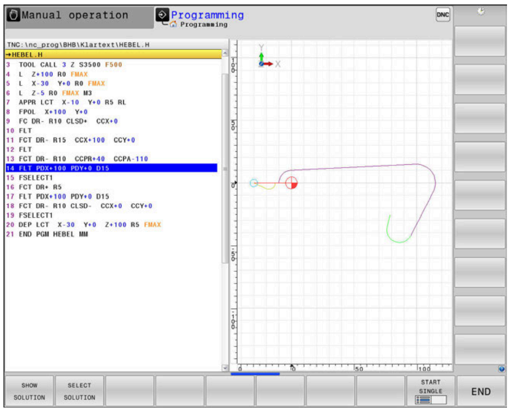
Display of the FK soft key depends on the spindle axis specified when defining the workpiece blank. If for example you enter spindle axis **Z** in the workpiece blank definition, the control only shows FK soft keys for the X/Y plane.

Switch the working plane

If you need a different working plane from the currently active plane, then proceed as follows:

- Press the **PLANE XY ZX YZ** soft key
- > The control then displays the FK soft keys in the newly selected plane.

FK programming graphics



> To use graphical support during FK programming, select the **PROGRAM + GRAPHICS** screen layout.
> **Further information:** "Programming", Page 72

> Program all of the contours before combining them (e.g., with the SL cycles). You thereby ensure that the contours are correctly defined and avoid unnecessary error messages.

Incomplete coordinate data often is not sufficient to fully define a workpiece contour. In this case, the control indicates the possible solutions in the FK graphic. You can then select the contour that matches the drawing.

The control uses various colors in the FK graphics:

- **blue:** uniquely specified contour element
  The last FK element is only shown in blue after the departure movement.
- **violet:** not yet uniquely specified contour element
- **ocher:** tool midpoint path
- **red:** rapid traverse
- **green:** more than one solution is possible

If the data permit several possible solutions and the contour element is displayed in green, select the correct contour element as follows:

SHOW SOLUTION

- Press the **SHOW SOLUTION** soft key repeatedly until the correct contour element is displayed. Use the zoom function if you cannot distinguish between possible solutions in the standard view

SELECT SOLUTION

- If the displayed contour element matches the drawing, then select this contour element with **SELECT SOLUTION**

If you do not yet wish to define a green contour element, then press the **START SINGLE** soft key to continue the FK dialog.

> You should define the green contour elements as soon as possible with **SELECT SOLUTION** to limit ambiguity for the subsequent contour elements.

Showing block numbers in the graphic window

To show a block number in the graphic window:

SHOW BLOCK NO. OFF ON

- Set the **SHOW BLOCK NO.** soft key to **ON**

Initiating the FK dialog

Proceed as follows to open the FK dialog:

FK

- Press the **FK** key
- > The control then displays the soft-key row with the FK functions.

If you initiate the FK dialog with one of these soft keys, the control shows additional soft-key rows. You can use them to enter known coordinates, directional data, and data regarding the course of the contour.

Soft key	FK element
FLT	Straight line with tangential connection
FL	Straight line without tangential connection
FCT	Circular arc with tangential connection
FC	Circular arc without tangential connection
FPOL	Pole for FK programming
PLANE XY ZX YZ	Select the working plane

Terminating the FK dialog

Proceed as follows to exit the soft-key row for FK programming:

END

- Press the **END** soft key

Alternative:

FK

- Press the **FK** key again

Pole for FK programming

FK

- To display the soft keys for free contour programming, press the **FK** key

FPOL

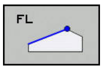
- To initiate the dialog for defining the pole, press the **FPOL** soft key
- > The control displays the axis soft keys of the active working plane.
- Enter the pole coordinates using these soft keys

> The pole for FK programming remains active until you define a new one using FPOL.

Free straight line programming

Straight line without tangential connection

- To display the soft keys for free contour programming, press the **FK** key

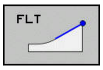


- To initiate the dialog for free programming of straight lines, press the **FL** soft key
- > The control displays additional soft keys.
- Enter all known data in the NC block by using these soft keys
- > The FK graphic displays the programmed contour element in violet until sufficient data is entered. If the entered data describes several solutions, the graphic will display the contour element in green.
  Further information: "FK programming graphics", Page 175

Straight line with tangential connection

If the straight line connects tangentially to another contour element, initiate the dialog with the **FLT** soft key:

- To display the soft keys for free contour programming, press the **FK** key

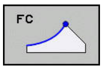


- To initiate the dialog, press the **FLT** soft key
- Enter all known data in the NC block by using the soft keys

Free circular path programming

Circular arc without tangential connection

- To display the soft keys for free contour programming, press the **FK** key

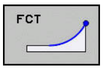


- To initiate the dialog for free programming of circular arcs, press the **FC** soft key
- > The control displays soft keys with which you can enter direct data on the circular arc or data on the circle center.
- Enter all known data in the NC block by using these soft keys
- > The FK graphic displays the programmed contour element in violet until sufficient data is entered. If the entered data describes several solutions, the graphic will display the contour element in green.
  Further information: "FK programming graphics", Page 175

Circular arc with tangential connection

If the circular arc connects tangentially to another contour element, initiate the dialog with the **FCT** soft key:

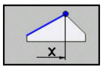
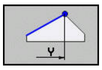
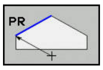
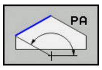
- To display the soft keys for free contour programming, press the **FK** key

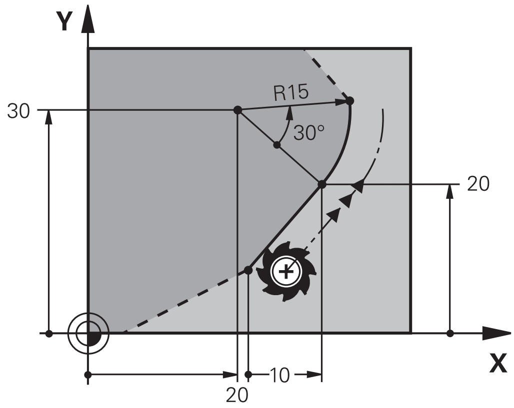


- To initiate the dialog, press the **FCT** soft key
- Enter all known data in the NC block by using the soft keys

Input possibilities

End point coordinates

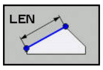
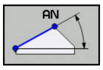
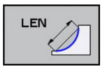
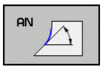
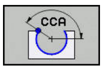
Soft keys		Known data
		Cartesian coordinates X and Y
		Polar coordinates referenced to FPOL

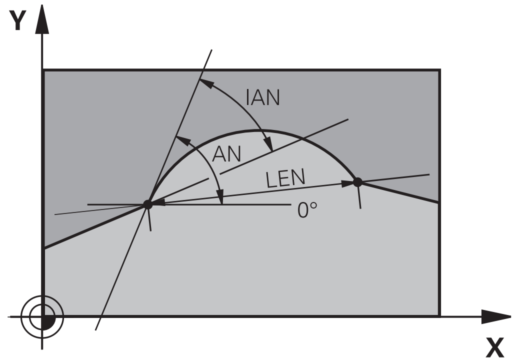


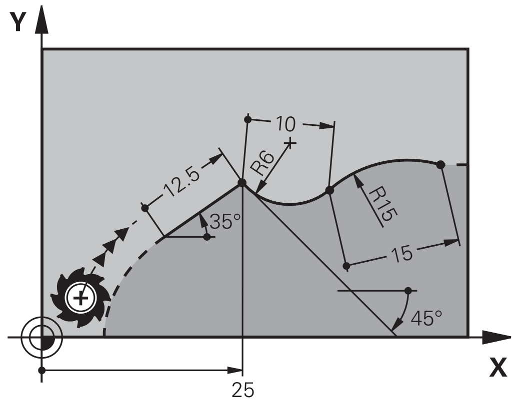
Example

```
7 FPOL X+20 Y+30
8 FL  IX+10 Y+20  RR F100
9 FCT PR+15 IPA+30  DR+ R15
```

Direction and length of contour elements

Soft keys	Known data
	Length of a straight line
	Gradient angle of a straight line
	Chord length LEN of an arc
	Gradient angle AN of an entry tangent
	Center angle of an arc





NOTICE

Danger of collision!

Incremental gradient angles **IAN** are referenced by the control to the direction of the previous traversing block. NC programs from previous control models (including iTNC 530) are not compatible. There is danger of collision during the execution of imported NC programs!

- Check the sequence and contour with the aid of the graphic simulation
- Adapt imported NC programs if required

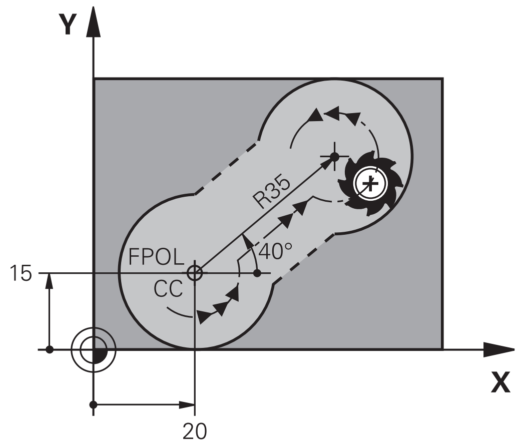
Example

```
27 FLT X+25 LEN 12.5  AN+35 RL  F200
28 FC  DR+ R6  LEN 10 AN-45
29 FCT  DR- R15  LEN 15
```

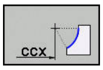
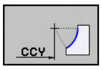
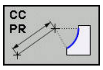
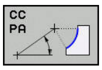
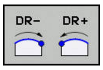
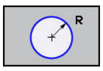
Circle center CC, radius and direction of rotation in the FC/FCT block

The control calculates a circle center for free-programmed arcs from the data you enter. This makes it possible to program full circles in an NC block with FK programming.

If you wish to define the circle center in polar coordinates you must use FPOL, not **CC**, to define the pole. FPOL is entered in Cartesian coordinates and remains in effect until the TNC encounters a NC block in which another **FPOL** is defined.



> A programmed or automatically calculated circle center or pole is effective only in connected conventional or FK sections. If an FK section splits up two conventionally programmed sections, the information about a circle center or pole will be lost. The two conventionally programmed sections must each have their own (if necessary, identical) CC blocks. Conversely, this information will also be lost if there is a conventional section between two FK sections.

Soft keys		Known data
		Circle center in Cartesian coordinates
		Center point in polar coordinates
		Rotational direction of the arc
		Radius of an arc

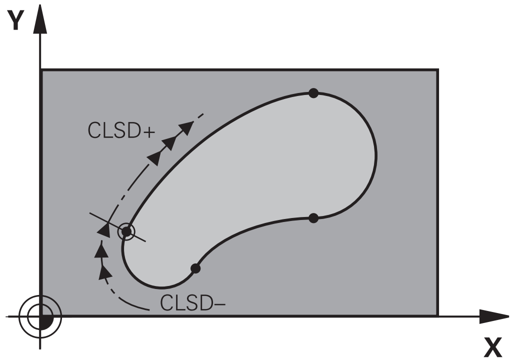
Example

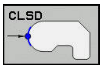
```
10 FC  CCX+20 CCY+15  DR+ R15
11 FPOL  X+20 Y+15
12 FL  AN+40
13 FC  DR+ R15  CCPR+35 CCPA+40
```

Closed contours

You can identify the beginning and end of a closed contour with the **CLSD** soft key. This reduces the number of possible solutions for the last contour element.

Enter **CLSD** as an addition to another contour data entry in the first and last NC blocks of an FK section.



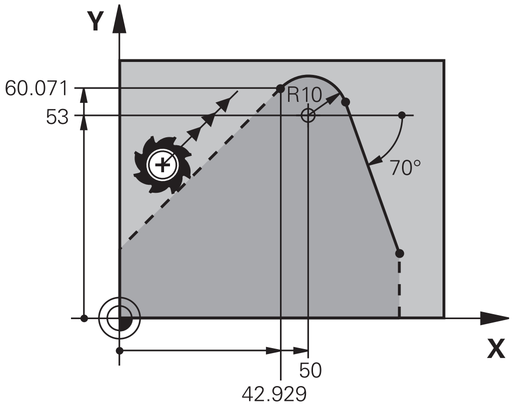
Soft key	Known data	
	Beginning of contour:	CLSD+
	End of contour:	CLSD–

Example

```
12 L  X+5 Y+35  RL F500  M3
13 FC  DR- R15  CLSD+ CCX+20  CCY+35
...
17 FC  DR- R+15  CLSD-
```

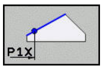
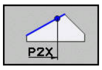
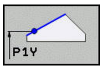
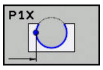
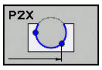
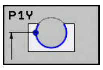
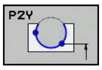
Auxiliary points

For both free-programmed straight lines and free-programmed circular arcs, you can enter the coordinates of auxiliary points that are located on the contour or in its proximity.

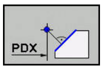
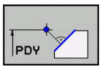
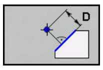
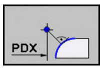
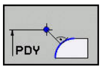
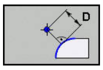


Auxiliary points on a contour

The auxiliary points are located on the straight line, the extension of the straight line, or on the circular arc.

Soft keys		Known data
		X coordinate of an auxiliary point P1 or P2 of a straight line
		Y coordinate of an auxiliary point P1 or P2 of a straight line
		X coordinate of an auxiliary point P1, P2 or P3 of a circular path
		Y coordinate of an auxiliary point P1, P2 or P3 of a circular path

Auxiliary points near a contour

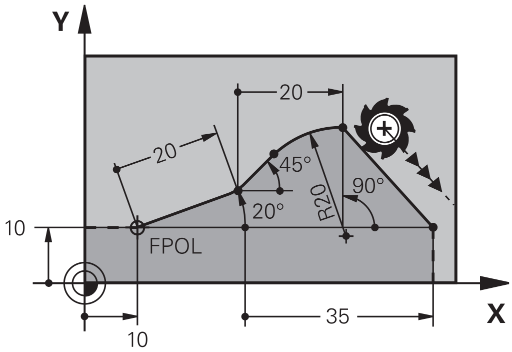
Soft keys		Known data
		X and Y coordinates of the auxiliary point near a straight line
		Distance of auxiliary point to straight line
		X and Y coordinates of an auxiliary point near a circular arc
		Distance of auxiliary point to circular arc

Example

```
13 FC  DR- R10  P1X+42.929 P1Y+60.071
14 FLT  AN-70 PDX+50  PDY+53 D10
```

Relative data

Relative data are values based on another contour element. The soft keys and program words for relative entries begin with the letter **R**. The figure on the right shows the dimensional data that should be programmed as relative data.



> The coordinates and angles for relative data are always programmed in incremental dimensions. You must also enter the NC block number of the contour element on which the data are based.
>
> The block number of the contour element on which the relative data are based can only be located up to 64 positioning blocks before the NC block in which you program the reference.
>
> If you delete an NC block on which relative data are based, the control will display an error message. Change the NC program first before you delete this NC block.

Data relative to NC block N: End point coordinates

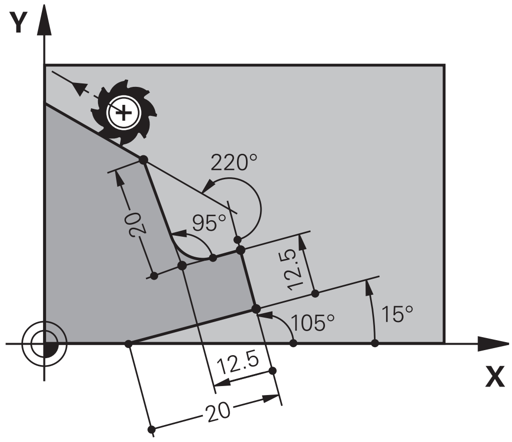
Soft keys		Known data
RX N...	RY N...	Cartesian coordinates relative to NC block N
RPR N...	RPA N...	Polar coordinates relative to NC block N

Example

```
12 FPOL X+10 Y+10
13 FL PR+20 PA+20
14 FL  AN+45
15 FCT IX+20 DR- R20 CCA+90 RX 13
16 FL IPR+35 PA+0 RPR 13
```

Data relative to NC block N: Direction and distance of the contour element

Soft key	Known data
RAN N...	Angle between a straight line and another element or between the entry tangent of the arc and another element
PAR N...	Straight line parallel to another contour element
DP	Distance from a straight line to a parallel contour element

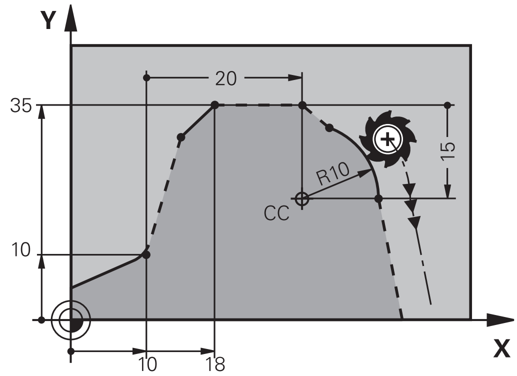


Example

```
17 FL  LEN 20 AN+15
18 FL AN+105 LEN 12.5
19 FL PAR 17 DP 12.5
20 FSELECT 2
21 FL  LEN 20 IAN+95
22 FL  IAN+220 RAN 18
```

Data relative to NC block N: Circle center CC

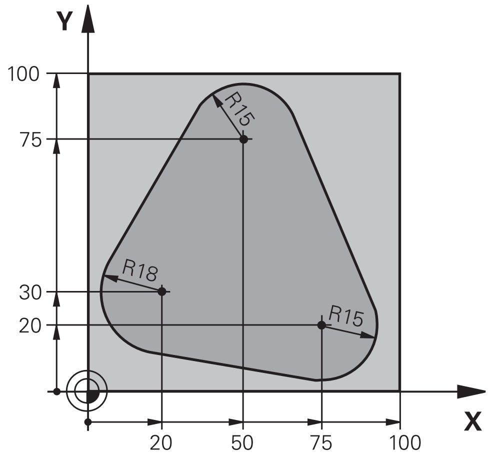
Soft key		Known data
RCCX N...	RCCY N...	Cartesian coordinates of the circle center relative to NC block N
RCCPR N...	RCCPA N...	Polar coordinates of the circle center relative to NC block N



Example

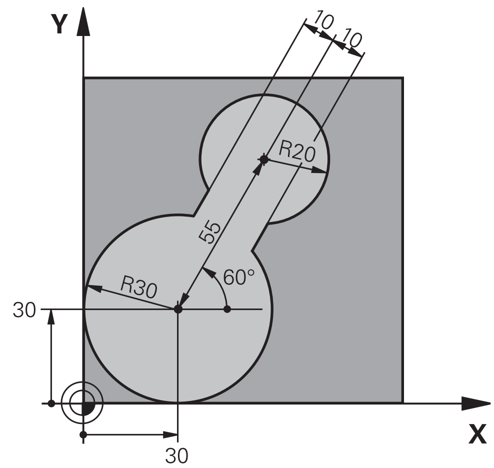
```
12 FL X+10 Y+10 RL
13 FL  ...
14 FL X+18 Y+35
15 FL  ...
16 FL  ...
17 FC DR- R10 CCA+0 ICCX+20 ICCY-15 RCCX12 RCCY14
```

Example: FK programming 1



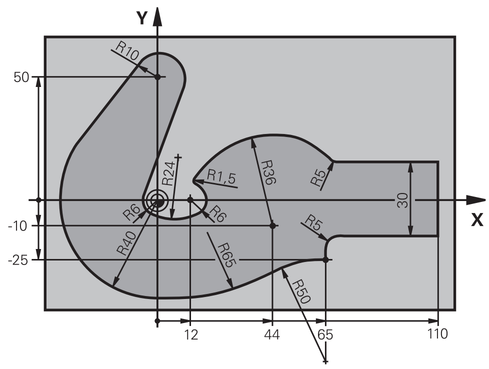
0 BEGIN PGM FK1 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	Workpiece blank definition
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 1 Z S500	Tool call
4 L Z+250 R0 FMAX	Retract the tool
5 L X-20 Y+30 R0 FMAX	Pre-position the tool
6 L Z-10 R0 F1000 M3	Move to working depth
7 APPR CT X+2 Y+30 CCA90 R+5 RL F250	Approach the contour on a circular arc with tangential connection
8 FC DR- R18 CLSD+ CCX+20 CCY+30	FK contour section:
9 FLT	Program all known data for each contour element
10 FCT DR- R15 CCX+50 CCY+75	
11 FLT	
12 FCT DR- R15 CCX+75 CCY+20	
13 FLT	
14 FCT DR- R18 CLSD- CCX+20 CCY+30	
15 DEP CT CCA90 R+5 F1000	Depart the contour on a circular arc with tangential connection
16 L X-30 Y+0 R0 FMAX	
17 L Z+250 R0 FMAX M2	Retract the tool, end of program
18 END PGM FK1 MM	

Example: FK programming 2



0 BEGIN PGM FK2 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	Workpiece blank definition
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 1 Z S4000	Tool call
4 L Z+250 R0 FMAX	Retract the tool
5 L X+30 Y+30 R0 FMAX	Pre-position the tool
6 L Z+5 R0 FMAX M3	Pre-position the tool
7 L Z-5 R0 F100	Move to working depth
8 APPR LCT X+0 Y+30 R5 RR F350	Approach the contour on a circular arc with tangential connection
9 FPOL X+30 Y+30	FK contour section:
10 FC DR- R30 CCX+30 CCY+30	Program all known data for each contour element
11 FL AN+60 PDX+30 PDY+30 D10	
12 FSELECT 3	
13 FC DR- R20 CCPR+55 CCPA+60	
14 FSELECT 2	
15 FL AN-120 PDX+30 PDY+30 D10	
16 FSELECT 3	
17 FC X+0 DR- R30 CCX+30 CCY+30	
18 FSELECT 2	
19 DEP LCT X+30 Y+30 R5	Depart the contour on a circular arc with tangential connection
20 L Z+250 R0 FMAX M2	Retract the tool, end of program
21 END PGM FK2 MM	

Example: FK programming 3



0 BEGIN PGM FK3 MM	
1 BLK FORM 0.1 Z X-45 Y-45 Z-20	Workpiece blank definition
2 BLK FORM 0.2 X+120 Y+70 Z+0	
3 TOOL CALL 1 Z S4500	Tool call
4 L Z+250 R0 FMAX	Retract the tool
5 L X-70 Y+0 R0 FMAX	Pre-position the tool
6 L Z-5 R0 F1000 M3	Move to working depth
7 APPR CT X-40 Y+0 CCA90 R+5 RL F250	Approach the contour on a circular arc with tangential connection
8 FC DR- R40 CCX+0 CCY+0	FK contour section:
9 FLT	Program all known data for each contour element
10 FCT DR- R10 CCX+0 CCY+50	
11 FLT	
12 FCT DR+ R6 CCX+0 CCY+0	
13 FCT DR+ R24	
14 FCT DR+ R6 CCX+12 CCY+0	
15 FSELECT 2	
16 FCT DR- R1.5	
17 FCT DR- R36 CCX+44 CCY-10	
18 FSELECT 2	
19 FCT DR+ R5	
20 FLT X+110 Y+15 AN+0	
21 FL AN-90	
22 FL X+65 AN+180 PAR21 DP30	
23 RND R5	
24 FL X+65 Y-25 AN-90	
25 FC DR+ R50 CCX+65 CCY-75	
26 FCT DR- R65	
27 FSELECT 1	
28 FCT Y+0 DR- R40 CCX+0 CCY+0	
29 FSELECT 4	

30 DEP CT CCA90 R+5 F1000	Depart the contour on a circular arc with tangential connection
31 L X-70 R0 FMAX	
32 L Z+250 R0 FMAX M2	Retract the tool, end of program
33 END PGM FK3 MM	

6

Programming Aids

6.1 GOTO function

Using the GOTO key

Jumping with the GOTO key

Use the **GOTO** key to jump to a specific location in the NC program, regardless of the active operating mode.

Proceed as follows:

GOTO

- Press the **GOTO** key
- > The control opens a pop-up window.
- Enter a number

N LINES ↓

- Select the jump statement by soft key, e.g. move down the number of lines entered

The control provides the following options:

Soft key	Function
N LINES ↑	Move up the number of lines entered
N LINES ↓	Move down the number of lines entered
GOTO LINE NUMBER	Jump to the block number entered

> Use the **GOTO** function only during programming and testing of NC programs. Use the block scan function during program run.
>
> **Further information:** User's Manual for Setup, Testing and Running NC Programs

Quick selection with the GOTO key

With the **GOTO** key, you can open the Smart Select window that makes it easy for you to select special functions or cycles.

Proceed as follows to select special functions:

SPEC FCT

- Press the **SPEC FCT** key

GOTO

- Press the **GOTO** key
- > The control displays a pop-up window showing a structural view of the special functions
- Select the desired function

Further information: Cycle Programming User's Manual

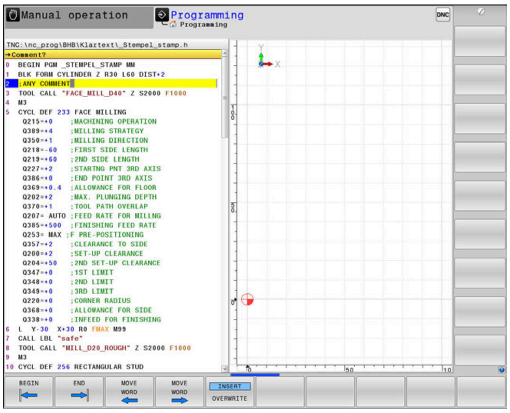
Opening the selection window with the GOTO key

When the control provides a selection menu, you can use the **GOTO** key to open the selection window. This allows you to view the available entries.

6.2 Display of NC programs

Syntax highlighting

The control displays syntax elements with various colors according to their meaning. Color-highlighting makes the NC programs easier to read and clearer.



Color highlighting of syntax elements

Use	Color
Standard color	Black
Display of comments	Green
Display of numerical values	Blue
Display of the block number	Violet
Display of FMAX	Orange
Display of the feed rate	Brown

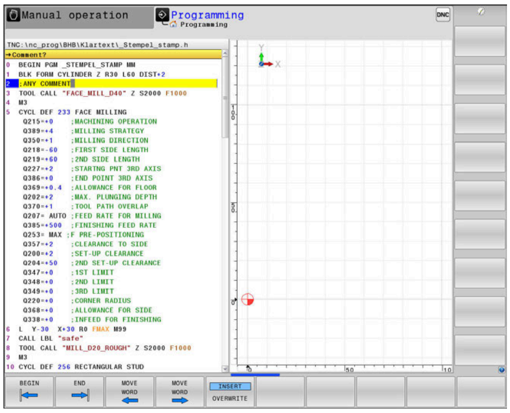
Scrollbar

Screen content can be shifted with the mouse using the scroll bar at the right edge of the program window. In addition, the size and position of the scrollbar indicates program length and cursor position.

6.3 Adding comments

Application

You can add comments to an NC program to explain program steps or make general notes.



> The control shows long comments in different ways, depending on the machine parameter **lineBreak** (no. 105404). It either wraps the comment lines or displays the **>>** symbol to indicate additional content.
>
> The last character in a comment block must not have any tilde(~).

You can add comments in different ways.

Entering comments during programming

- Enter the data for an NC block
- Press the semicolon key **;** on the alphabetic keyboard
- \> The control displays the dialog prompt **Comment?**
- Enter the comment
- Press the **END** key to conclude the NC block

Inserting comments after program entry

- Select the NC block to which you want to add the comment
- Select the last word in the NC block with the right arrow key:
- Press the semicolon key **;** on the alphabetic keyboard
- \> The control displays the dialog prompt **Comment?**
- Enter the comment
- Press the **END** key to conclude the NC block

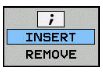
Entering a comment in a separate NC block

- Select the NC block after which you want to insert the comment
- Initiate the programming dialog with the semicolon key (**;**) on the alphabetic keyboard
- Enter your comment and conclude the NC block by pressing the **END** key

Commenting out an existing NC block

Proceed as follows to change an existing NC block to a comment:

- Select the NC block to be commented out



- Press the **INSERT COMMENT** soft key

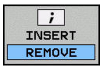
Alternative:

- Press the **<** key on the alphabetic keyboard
- \> The control inserts a semicolon **;** at the beginning of the block.
- Press the **END** key

Changing a comment for an NC block

Proceed as follows to change a commented-out NC block to an active NC block:

- Select the comment block you want to change

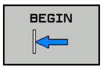
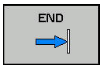
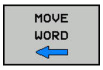
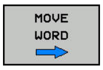
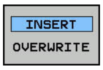


- Press the **REMOVE COMMENT** soft key

Alternative:

- Press the **>** key on the alphabetic keyboard
- > The control removes the semicolon **;** at the beginning of the block.
- Press the **END** key

Functions for editing a comment

Soft key	Function
	Jump to beginning of comment
	Jump to end of comment
	Jump to the beginning of a word. Use a space to separate words
	Jump to the end of a word. Use a space to separate words
	Switch between paste and overwrite mode

6.4 Freely editing an NC program

Certain syntax elements, such as LN blocks, cannot be entered directly in the NC editor by using the available keys and soft keys.

To prevent the use of an external text editor, the control offers the following possibilities:

- Free syntax input using the control's integrated text editor
- Free syntax input using the **?** key in the NC editor

Free syntax input using the control's integrated text editor

Proceed as follows to add syntax to an existing NC program:

PGM MGT

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

MORE FUNCTIONS

- Press the **MORE FUNCTIONS** soft key

SELECT EDITOR

- Press the **SELECT EDITOR** soft key
- > The control opens a selection window.

OK

- Select the **TEXT EDITOR** option
- Confirm your selection with **OK**
- Add the desired syntax

> The control does not check the syntax in the text editor. Check your entries in the NC editor when you are finished.

Free syntax input using the ? key in the NC editor

Proceed as follows to add syntax to an existing, open NC program:

⇧

?
/

- Enter **?**
- > The control opens a new NC block.

END
□

- Add the desired syntax
- Confirm your entry with **END**

> After confirmation, the control checks the syntax. Errors will result in **ERROR** blocks.

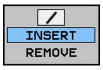
6.5 Skipping NC blocks

Insert a slash (/)

You can optionally hide NC blocks.

Proceed as follows in order to hide NC blocks in the **Programming** mode:

- Select the desired NC block

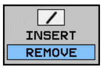


- Press the **INSERT** soft key
- > The control inserts a slash (/).

Delete the slash (/)

Proceed as follows in order to show NC blocks again in the **Programming** mode:

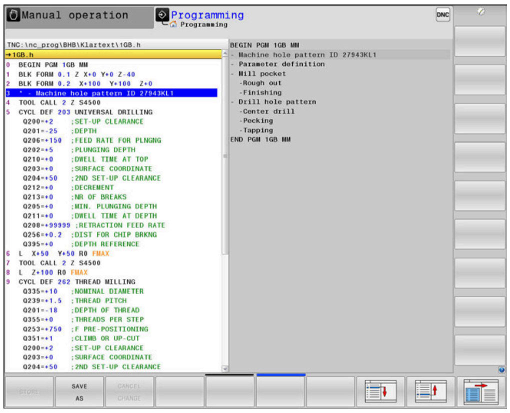
- Select the hidden NC block



- Press the **REMOVE** soft key
- > The control removes the slash (/).

6.6 Structuring NC programs

Definition and applications



The control enables you to comment NC programs in structuring blocks. Structuring blocks are texts with up to 252 characters and are used as comments or headlines for the subsequent program lines.

With the aid of appropriate structuring blocks, you can organize long and complex NC programs in a clear and comprehensible manner.

This function is particularly convenient if you want to change the NC program later. Structuring blocks can be inserted into the NC program at any point.

Structure blocks can also be displayed in a separate window, and be edited or added to, as desired. Use the appropriate screen layout for this.

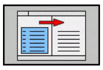
The control manages the inserted structure items in a separate file (extension: .SEC.DEP). This speeds navigation in the program structure window.

The **PROGRAM + SECTS** screen layout can be selected in the following operating modes:

- **Program run, single block**
- **Program run, full sequence**
- **Programming**

Displaying the program structure window / Changing the active window

- Display structure window: For this screen layout press the **PROGRAM + SECTS** soft key



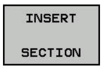
- Change the active window: Press the **CHANGE WINDOW** soft key

Inserting a structure block in the program window

- Select the NC block after which you want to insert the structuring block

- Press the **SPEC FCT** key

- Press the **PROGRAMMING AIDS** soft key



- Press the **INSERT SECTION** soft key
- Enter the structuring text

- Change the structuring depth (indenting) via soft key

> You can indent structure items only during editing.

> You can also insert structure blocks with the key combination **Shift + 8**.

Selecting blocks in the program structure window

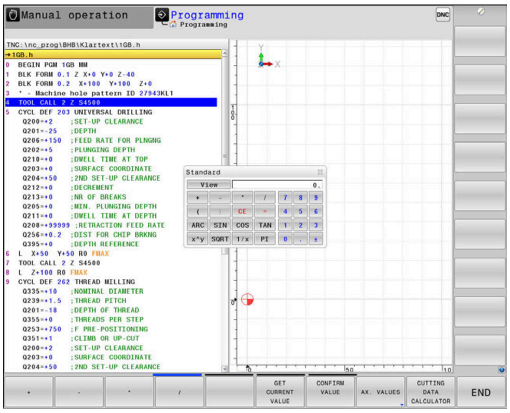
If you are scrolling through the program structure window block by block, the control at the same time automatically moves the corresponding NC blocks in the program window. This way you can quickly skip large program sections.

6.7 Calculator

Operation

The control features a calculator with the most important mathematical functions.

- To show the calculator, press the **CALC** key
- Select the arithmetic functions: Select the command via soft key or enter it with an alphanumeric keyboard
- To close the calculator, press the **CALC** key



Calculate function	Command (soft key)
Addition	+
Subtraction	–
Multiplication	*
Division	/
Calculating with parentheses	( )
Arc cosine	ARC
Sine	SIN
Cosine	COS
Tangent	TAN
Exponent	X^Y
Square root	SQRT
Inverted function	1/x
PI (3.14159265359)	PI
Add value to buffer memory	M+
Save value to buffer memory	MS
Retrieve buffer memory contents	MR
Delete buffer memory contents	MC
Natural logarithm	LN
Logarithm	LOG
Exponential function	e^x
Check the algebraic sign	SGN
Calculate the absolute value	ABS

Calculate function	Command (soft key)
Truncate decimal places	INT
Truncate digits before the decimal point	FRAC
Modulo	MOD
Select view	View
Delete value	CE
Unit of measure	MM or INCH
Show angular value in radians (default: angular value in degrees)	RAD
Select numerical value notation	DEC (decimal) or HEX (hexadecimal)

Transferring the calculated value into the NC program

- ▶ With the arrow keys, select the word into which the calculated value is to be transferred
- ▶ Show the calculator by pressing the **CALC** key, and perform the desired calculation
- ▶ Press the **CONFIRM VALUE** soft key
- > The control transfers the value into the active input field and closes the calculator.

> You can also transfer values from an NC program into the calculator. When you press the **GET CURRENT VALUE** soft key or the **GOTO** key, the control transfers the value from the active input field to the calculator.
>
> The calculator remains in effect even after a change in operating modes. Press the **END** soft key to close the calculator.

Functions in the pocket calculator

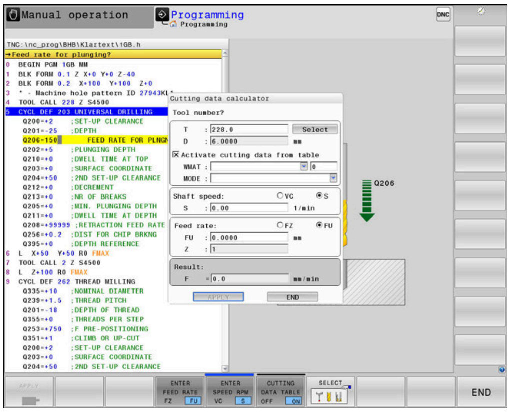
Soft key	Function
AX. VALUES	Transfer the nominal or reference value of the respective axis position into the calculator
GET CURRENT VALUE	Transfer the numerical value from the active input field into the calculator
CONFIRM VALUE	Transfer the numerical value from the calculator into the active input field
COPY FIELD	Copy the numerical value from the calculator
PASTE FIELD	Insert the copied numerical value into the calculator
CUTTING DATA CALCULATOR	Open the cutting data calculator

You can also move the calculator with the arrow keys of your alphabetic keyboard. If you have connected a mouse you can also position the calculator with this.

6.8 Cutting data calculator

Application

With the cutting data calculator you can calculate the spindle speed and the feed rate for a machining process. Then you can load the calculated values into an opened feed rate or spindle speed dialog box in the NC program.



> You cannot perform any cutting data calculation in turning mode with the cutting data calculator because the feed rate and spindle speed data are different in turning mode from milling mode.
>
> Feed rates in turning operations are often defined in millimeters per revolution (mm/1) (**M136**), whereas the cutting data calculator always calculates feed rates in millimeters per minute (mm/min). Furthermore, the radius in the cutting data calculator is referenced to the tool; turning operations, however, require the workpiece diameter.

To open the cutting data calculator, press the **CUTTING DATA CALCULATOR** soft key.

The control shows the soft key if you

- press the **CALC** key
- Define spindle speeds
- Define feed rates
- Press the **F** soft key in **Manual Operation** mode
- Press the **S** soft key in **Manual Operation** mode

Display modes of the cutting data calculator

The cutting data calculator is displayed with different input fields depending on whether you calculate a spindle speed or a feed rate:

Window for spindle speed calculation:

Abbrev.	Meaning
T:	Tool number
D:	Diameter of the tool
VC:	Cutting speed
S=	Result for spindle speed

If you open the speed calculator in a dialog where the tool is already defined, the speed calculator automatically applies the tool number and diameter. You only need to enter **VC** in the dialog field.

Window for feed rate calculation:

Abbrev.	Meaning
T:	Tool number
D:	Diameter of the tool
VC:	Cutting speed
S:	Spindle speed
Z:	Number of teeth
FZ:	Feed per tooth
FU:	Feed per revolution
F=	Result for feed rate

> You can transfer the feed rate from the **TOOL CALL** block into subsequent NC blocks by pressing the **F AUTO** soft key. If you have to change the feed rate later, you only need to adjust the feed rate value in the **TOOL CALL** block.

Functions of the cutting data calculator

You have the following possibilities depending on where you open the cutting data calculator:

Soft key	Function
APPLY	Transfer the value from the cutting data calculator into the NC program
CALCULATE FEEDRATE F SPEED S	Toggle between feed-rate calculation and spindle-speed calculation
ENTER FEED RATE FZ FU	Toggle between feed per tooth and feed per revolution
ENTER SPEED RPM VC S	Toggle between spindle speed and cutting speed
CUTTING DATA TABLE OFF ON	Activate or deactivate working with cutting data tables
SELECT	Select a tool from the tool table
↓	Move the cutting data calculator in the direction of the arrow
POCKET CALCULATOR	Switch to the calculator
INCH	Use inch values in the cutting data calculator
END	Close the cutting data calculator

Working with cutting data tables

Application

If you store tables for materials, cutting materials, and cutting data on the control, then the cutting data calculator can use the values in these tables.

Proceed as follows before working with automatic calculation of the spindle speed and feed rate:

- Enter the type of workpiece material in the table WMAT.tab
- Enter the type of cutting material in the file TMAT.tab
- Enter the combination of workpiece material and cutting material in a cutting data table
- Define the tool with the necessary values in the tool table
  - Tool radius
  - Number of teeth
  - Cutting material
  - Cutting data table

Workpiece material WMAT

Define the workpiece materials in the WMAT.tab table. You must save this table in the directory **TNC:\table**.

This table contains the column **WMAT** for the material and a column called **MAT_CLASS**; here you categorize the materials into material classes with the same cutting conditions, e.g. according to DIN EN 10027-2.

Enter the workpiece material as follows in the cutting data calculator:

- Select the cutting data calculator
- Select **Activate cutting data from table** in the pop-up window
- Select **WMAT** from the drop-down menu

TNC:\table\WMAT.TAB

NR	WMAT	MAT_CLASS
1		10
2	1.0038	10
3	1.0044	10
4	1.0114	10
5	1.0177	10
6	1.0143	10
7	St 37-2	10
8	St 37-3 N	10
9	X 14 CrMo S 17	20
10	1.1404	20
11	1.4305	20
12	V2A	21
13	1.4301	21
14	AlCu4PBMg	100
15	Aluminium	100
16	PTFE	200

Cutting material TMAT

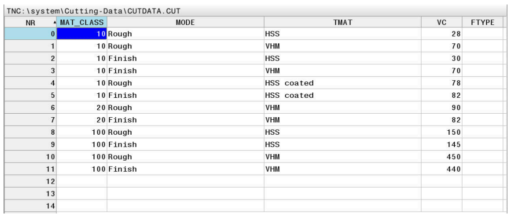
Cutting materials are defined in the TMAT.tab table. You must save this table in the directory **TNC:\table**.

You assign the cutting material in the **TMAT** column of the tool table. You can create columns with other names, such as **ALIAS1** and **ALIAS2** in order to enter alternative names for the same cutting material.

Cutting data table

Define the combinations of workpiece material and cutting material with the corresponding cutting data in a table with the file extension .CUT. You must save this table in the directory **TNC:\system\Cutting-Data**.

You assign the appropriate cutting data table in the **CUTDATA** column of the tool table.



TNC:\system\Cutting-Data\CUTDATA.CUT

NR	MAT_CLASS	MODE	TMAT	VC	FTYPE
0	10	Rough	HSS	28	
1	10	Rough	VHM	70	
2	10	Finish	HSS	30	
3	10	Finish	VHM	70	
4	10	Rough	HSS coated	78	
5	10	Finish	HSS coated	82	
6	20	Rough	VHM	90	
7	20	Finish	VHM	82	
8	100	Rough	HSS	150	
9	100	Finish	HSS	145	
10	100	Rough	VHM	450	
11	100	Finish	VHM	440	
12					
13					
14					

> Use this simplified table if you use tools that have only a single diameter, or if the diameter is not relevant to the feed rate, i.e. for indexable inserts.

The cutting data table contains the following columns:

- **MAT_CLASS**: Material class
- **MODE**: Machining mode, such as finishing
- **TMAT**: Cutting material
- **VC**: Cutting speed
- **FTYPE**: Type of feed rate **FZ** or **FU**
- **F**: Feed rate

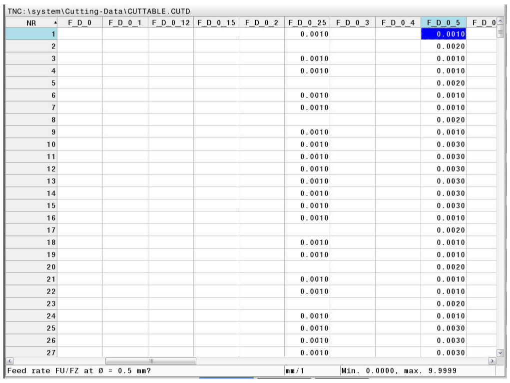
Diameter-dependent cutting data table

In many cases the diameter of the tool determines which cutting data you can use. Use the cutting data table with the file extension .CUTD for this purpose. You must save this table in the directory **TNC:\system\Cutting-Data**.

You assign the appropriate cutting data table in the **CUTDATA** column of the tool table.

The diameter-dependent cutting data table contains the following additional columns:

- **F_D_0**: Feed rate for Ø 0 mm
- **F_D_0_1**: Feed rate for Ø 0.1 mm
- **F_D_0_12**: Feed rate for Ø 0.12 mm
- ...



TNC:\system\Cutting-Data\CUTTABLE.CUTD

NR	F_D_0	F_D_0_1	F_D_0_12	F_D_0_15	F_D_0_2	F_D_0_25	F_D_0_3	F_D_0_4	F_D_0_5	F_D_0
1						0.0010			0.0010	
2									0.0020	
3						0.0010			0.0010	
4						0.0010			0.0010	
5									0.0020	
6						0.0010			0.0010	
7						0.0010			0.0010	
8									0.0020	
9						0.0010			0.0010	
10						0.0010			0.0030	
11						0.0010			0.0030	
12						0.0010			0.0030	
13						0.0010			0.0030	
14						0.0010			0.0030	
15						0.0010			0.0030	
16						0.0010			0.0010	
17									0.0020	
18						0.0010			0.0010	
19						0.0010			0.0010	
20									0.0020	
21						0.0010			0.0010	
22						0.0010			0.0010	
23									0.0020	
24						0.0010			0.0010	
25						0.0010			0.0030	
26						0.0010			0.0030	
27						0.0010			0.0030	

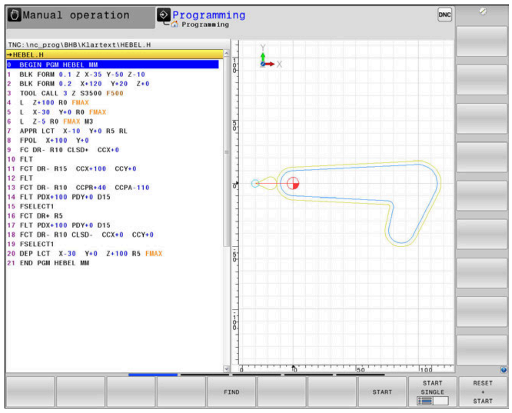
Feed rate FU/FZ at Ø = 0.5 mm? | mm/1 | Min. 0.0000, max. 9.9999

> You don't need to fill in all columns. If a tool diameter is between two defined columns, the control linearly interpolates the feed rate.

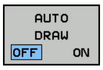
6.9 Programming graphics

Activating and deactivating programming graphics

While you are writing an NC program, you can have the control generate a 2-D pencil-trace graphic of the programmed contour.



- Press the **Screen layout** key
- Press the **PROGRAM + GRAPHICS** soft key
- > The control shows the NC program to the left and graphics to the right.



- Set the **AUTO DRAW** soft key to **ON**
- > While you are entering the program lines, the control generates each programmed movement in the graphics window in the right screen half.

If you do not want the control to generate graphics during programming, then set the **AUTO DRAW** soft key to **OFF**.

If **AUTO DRAW** is set to **ON**, then the control ignores the following program content when creating 2-D pencil-trace graphics:
- Program section repetitions
- Jump commands
- M functions, such as M2 or M30
- Cycle calls
- Warnings due to locked tools

Therefore, only use automatic drawing during contour programming.

The control resets the tool data when you reopen an NC program or press the **RESET + START** soft key.

The control uses various colors in the programming graphics:
- **blue:** uniquely specified contour element
- **violet:** not yet uniquely specified contour element, can still be modified by e.g. an RND
- **light blue:** holes and threads
- **ocher:** tool midpoint path
- **red:** rapid traverse

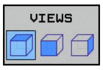
Further information: "FK programming graphics", Page 175

Generating a graphic for an existing NC program

- Use the arrow keys to select the NC block up to which you want the graphic to be generated, or press **GOTO** and enter the desired block number

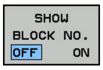
- Reset previously active tool data, and generate graphic: Press the **RESET + START** soft key

Additional functions:

Soft key	Function
	Reset previously active tool data. Generate programming graphics
	Generate programming graphic blockwise
	Generate a complete programming graphic, or complete it after **RESET + START**
	Stop the programming graphics. This soft key only appears while the control is generating the programming graphics
	Selecting views ■ Plan view ■ Front view ■ Page view
	Display or hide tool paths
	Display or hide tool paths in rapid traverse

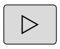
Block number display ON/OFF

- Shift the soft-key row



- Show block numbers: Set the **SHOW BLOCK NO.** soft key to **ON**
- Hide block numbers: Set the **SHOW BLOCK NO.** soft key to **OFF**

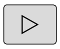
Erasing the graphic



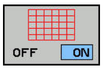
- Shift the soft-key row

- Erase the graphic: Press the **CLEAR GRAPHICS** soft key

Showing grid lines



- Shift the soft-key row

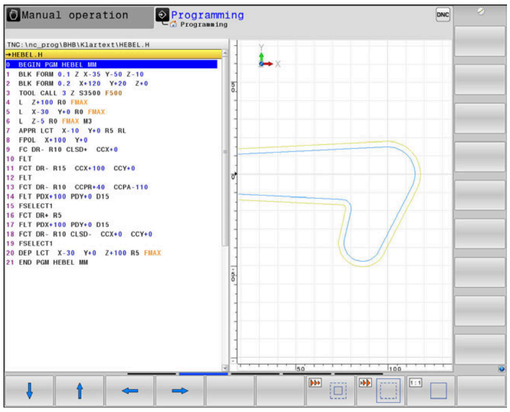


- Show grid lines: Press the **Show grid lines** soft key

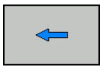
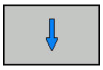
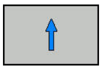
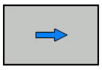
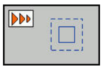
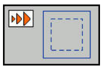
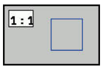
Magnification or reduction of details

You can select the graphics display

- Shift the soft-key row



The following functions are available:

Soft key		Function
 	 	Shift section
		Reduce section
		Enlarge section
		Reset section

The **RESET BLK FORM** soft key allows you to restore the original section.

You can also use the mouse to change the graphic display. The following functions are available:

- To shift the displayed model, hold down the center mouse button or the mouse wheel, and move the mouse. If you press the shift key at the same time, then you will be able to shift the model only horizontally or vertically.
- To zoom in on a certain area, mark a zoom area by holding the left mouse button down. After you release the left mouse button, the control zooms in on the defined area.
- To rapidly magnify or reduce any area, rotate the mouse wheel backwards or forwards.

6.10 Error messages

Display of errors

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

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

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

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

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

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

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

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

Opening the error window

ERR

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

Closing the error window

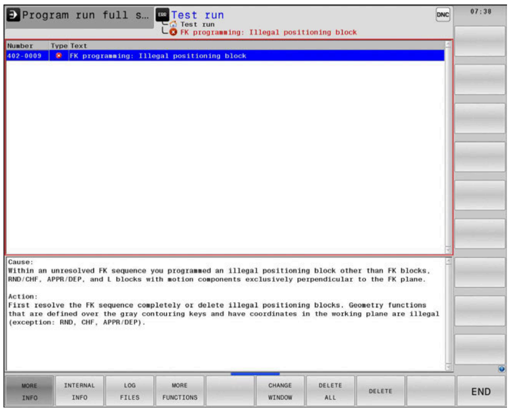
END

- Press the **END** soft key**END**

ERR

- Alternative: Press the **ERR** key
- > The control closes the error window.

Detailed error messages



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

- ▶ Open the error window

- ▶ Information on the error cause and corrective action: Position the cursor on the error message and press the **MORE INFO** soft key
- > The control opens a window with information on the error cause and corrective action.
- ▶ Leave Info: Press the **MORE INFO** soft key again

Soft key: INTERNAL INFO

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

- ▶ Open the error window

- ▶ Detailed information about the error message: Position the cursor on the error message and press the **INTERNAL INFO** soft key
- > The control opens a window with internal information about the error.
- ▶ Exit the details: Press the **INTERNAL INFO** soft key again

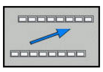
Soft key FILTER

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

- ▶ Open the error window

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

- ▶ Press the **FILTER** soft key
- > The control filters the identical warnings.



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

ACTIVATE AUTOMATIC SAVING soft key

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

- Open the error window

MORE FUNCTIONS

- Press the **MORE FUNCTIONS** soft key

ACTIVATE AUTOMATIC SAVING

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

STORE

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

Clearing errors

Clearing errors automatically

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

The factory default setting of the control defines that warning and error messages in the **Test Run** and **Programming** operating modes will be cleared automatically from the error window. Messages issued in the machine operating modes will not be cleared.

Clearing errors outside of the error window

CE

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

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

Clearing errors

- Open the error window

DELETE

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

DELETE ALL

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

> **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. The capacity of the error log is limited. If the log is full, the control uses a second file. If this is also full, the first error log is deleted and newly written etc. If required, switch from **CURRENT FILE** to **PREVIOUS FILE** to view the history.

- Open the error window.

LOG FILES

- Press the **LOG FILES** soft key

ERROR LOG

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

PREVIOUS FILE

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

CURRENT FILE

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

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

Keystroke log

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

LOG FILES

- Press the **LOG FILES** soft key

KEYSTROKE LOG

- Open the keystroke log file: Press the **KEYSTROKE LOG** soft key

PREVIOUS FILE

- Set the previous keystroke log if required: Press the **PREVIOUS FILE** soft key

CURRENT FILE

- Set the current keystroke log if required: Press the **CURRENT FILE** soft key

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

Overview of the keys and soft keys for viewing the log

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

Informational texts

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

Saving service files

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

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

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

Saving service files

- Open the error window

LOG FILES

- Press the **LOG FILES** soft key

SAVE SERVICE FILES

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

OK

- Save the service files: Press the **OK** soft key

Calling the TNCguide help system

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

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

HEIDENHAIN TNCguide

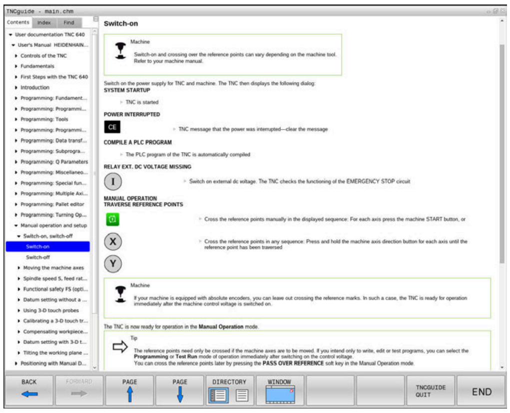
- Call the help for HEIDENHAIN error messages

MACHINE MFR.

- Call the help for HEIDENHAIN machine-specific error messages, if available

6.11 TNCguide context-sensitive help system

Application



> Before you can use the TNCguide, you need to download the help files from the HEIDENHAIN home page
>
> **Further information:** "Downloading current help files", Page 221

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

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

The following user documentation is available in TNCguide:

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

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

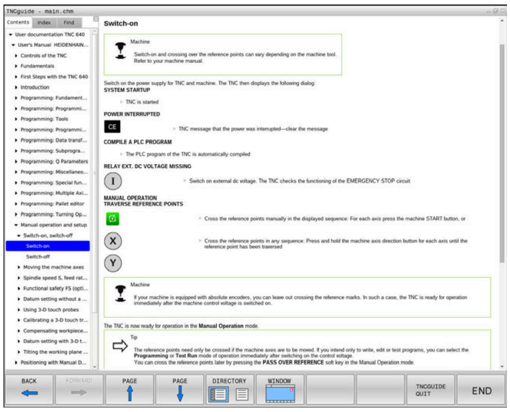
> As an option, your machine tool builder can embed machine-specific documentation in the **TNCguide**. These documents then appear as a separate book in the **main.chm** file.

Working with TNCguide

Calling TNCguide

There are several ways to start the TNCguide:

- Press the **HELP** key.
- Click the help symbol at the lower right of the screen beforehand, then click the appropriate soft keys
- Open a help file (CHM file) via the file management. The control can open any .chm file, even if it is not saved in the control's internal memory



> **i** On the Windows programming station, the TNCguide is opened in the internally defined standard browser.

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

- Select the soft-key row containing the desired soft key
- Click with the mouse on the help symbol that the control displays just above the soft-key row
- \> The mouse pointer turns into a question mark.
- Move the question mark to the soft key for which you want an explanation
- \> The control opens the TNCguide. If there is no entry point for the selected soft key, then the control opens the book file **main.chm**. You can search for the desired explanation using full text search or by using the navigation.

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

- Select any NC block
- Select the desired word
- Press the **HELP** key.
- \> The control opens the Help system and shows the description of the active function. This does not apply for miscellaneous functions or cycles from your machine manufacturer.

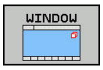
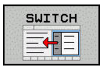
Navigating in the TNCguide

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

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

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

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

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

Subject index

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

The left side is active.

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

Alternative:

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

Full-text search

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

The left side is active.

- Select the **Find** tab
- Activate the **Find:** entry field
- Enter the search word
- Press the **ENT** key
- > The control lists all sources containing the word.
- Use the arrow keys to navigate to the desired source
- Press the **ENT** key to go to the selected source

> The full-text search only works for single words.
>
> If you activate the **Search only in titles** function, the control searches only through headings and ignores the body text. To activate the function, use the mouse or select it and then press the space bar to confirm.

Downloading current help files

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

Navigate to the suitable help file as follows:

- TNC Controls
- Series, e.g. TNC 600
- Desired NC software number, e.g.TNC 640 (34059x-10)
- Select the desired language version from the **TNCguide online help** table
- Download the ZIP file
- Extract the ZIP file
- Move the extracted CHM files to the **TNC:\tncguide\en** directory or the respective language subdirectory on the control

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

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

7

Miscellaneous Functions

7.1 Entering miscellaneous functions M and STOP

Fundamentals

With the control's miscellaneous functions—also called M functions—you can affect:

- the program run, e.g. a program interruption
- the machine functions, such as switching spindle rotation and coolant supply on and off
- The path behavior of the tool

You can enter up to four M (miscellaneous) functions at the end of a positioning block or in a separate NC block. The control displays the following dialog question: **Miscellaneous function M ?**

You usually enter only the number of the miscellaneous function in the programming dialog. With some miscellaneous functions, the dialog is extended so that you can enter the required parameters for this function.

In the **Manual operation** and **Electronic handwheel** operating modes, the M functions are entered with the **M** soft key.

Effectiveness of miscellaneous functions

Please note that some M functions become effective at the start of a positioning block, and others at the end, regardless of their position in the NC block.

Miscellaneous functions come into effect in the NC block in which they are called.

Some miscellaneous functions are effective only in the NC block in which they are programmed. Unless the miscellaneous function is only effective blockwise, you must either cancel it in a subsequent NC block with a separate M function, or it is automatically canceled by the control at the end of the program.

> If multiple functions were programmed in a single NC block, the execution sequence is as follows:
> - M functions taking effect at the start of the block are executed before those taking effect at the end of the block
> - If all M functions are effective at the start or end of the block, execution takes place in the sequence as programmed

Entering a miscellaneous function in a STOP block

If you program a **STOP** block, the program run or test run is interrupted at the block, e.g. for a tool inspection. You can also enter an M (miscellaneous) function in a **STOP** block:

STOP

- To program an interruption of program run, press the **STOP** key
- Enter a miscellaneous function **M** if required

Example

```
87 STOP
```

7.2 Miscellaneous functions for program run inspection, spindle and coolant

Overview

> Refer to your machine manual!
>
> The machine manufacturer can influence the behavior of the miscellaneous functions described below.

M	Effect	Effective at block	Start	End
M0	Program STOP Spindle STOP			■
M1	Optional program STOP Spindle STOP if necessary Coolant OFF if necessary (function defined by the machine tool builder)			■
M2	STOP program run Spindle STOP Coolant off Return jump to block 1 Clear status display Functional scope depends on machine parameter **resetAt** (no. 100901)			■
M3	Spindle ON clockwise		■	
M4	Spindle ON counterclockwise		■	
M5	Spindle STOP			■
M6	Tool change Spindle STOP Program STOP			■

> Since this function varies depending on the machine tool builder, HEIDENHAIN recommends that you use the **TOOL CALL** function for tool changes.

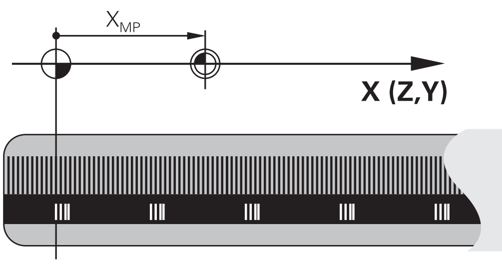
M	Effect	Effective at block	Start	End
M8	Coolant ON		■	
M9	Coolant OFF			■
M13	Spindle ON clockwise Coolant ON		■	
M14	Spindle ON counterclockwise Coolant ON		■	
M30	Same as M2			■

7.3 Miscellaneous functions for coordinate entries

Programming machine-referenced coordinates: M91/ M92

Scale datum

On the scale, a reference mark indicates the position of the scale datum.



Machine datum

The machine datum is required for the following tasks:

- Define the axis traverse limits (software limit switches)
- Approach machine-referenced positions (such as tool change positions)
- Set a workpiece preset

The distance in each axis from the scale datum to the machine datum is defined by the machine tool builder in a machine parameter.

Standard behavior

The control references the coordinates to the workpiece datum.

Further information: User's Manual for Setup, Testing and Running NC Programs

Behavior with M91—Machine datum

If you want the coordinates in a positioning block to reference the machine datum, enter M91 into these NC blocks.

> If you program incremental coordinates in an M91 block, enter them with respect to the last programmed M91 position. If no M91 position is programmed in the active NC block, then enter the coordinates with respect to the current tool position.

The coordinate values on the control's screen are referenced to the machine datum. Switch the display of coordinates in the status display to REF.

Further information: User's Manual for Setup, Testing and Running NC Programs

Behavior with M92 – Additional machine reference point

Refer to your machine manual!

In addition to the machine datum, the machine tool builder can also define an additional machine-based position as a machine reference point.

For each axis, the machine tool builder defines the distance between the machine reference point and the machine datum.

If you want the coordinates in positioning blocks to be based on the machine preset, enter M92 into these NC blocks.

Radius compensation remains the same in blocks that are programmed with **M91** or **M92**. The tool length will **not** be taken into account.

Effect

M91 and M92 are effective only in the blocks in which M91 and M92 have been programmed.

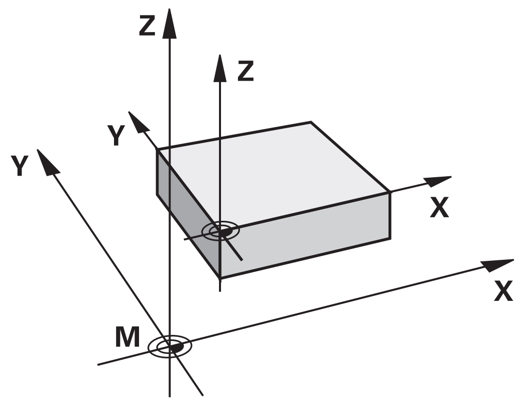
M91 and M92 take effect at the start of block.

Workpiece preset

If you want the coordinates to always reference to the machine datum, you can inhibit presetting for one or more axes.

If presetting is inhibited for all axes, the control no longer displays the **SET PRESET** soft key in the **Manual operation** mode.

The figure shows coordinate systems with the machine and workpiece datum.



M91/M92 in the Test Run mode

In order to be able to graphically simulate M91/M92 movements, you need to activate working space monitoring and display the workpiece blank referenced to the defined preset.

Further information: User's Manual for Setup, Testing and Running NC Programs

Moving to positions in a non-tilted coordinate system with a tilted working plane: M130

Standard behavior with a tilted working plane

The control references the coordinates in the positioning blocks to the tilted working plane coordinate system.

Behavior with M130

Despite an active tilted working plane, the control references the coordinates in straight line blocks to the non-tilted workpiece coordinate system.

The control then positions the tilted tool at the programmed coordinates of the non-tilted workpiece coordinate system.

NOTICE

Danger of collision!

The **M130** function is only active blockwise. The control executes the subsequent machining operations in the tilted working plane coordinate system again. Danger of collision during machining!

- Check the sequence and positions using a graphic simulation

Programming notes:

- The **M130** function is only allowed if the **Tilt the working plane** function is active.
- If the **M130** function is combined with a cycle call, the control will interrupt the execution with an error message.

Effect

M130 functions blockwise in straight-line blocks without tool radius compensation.

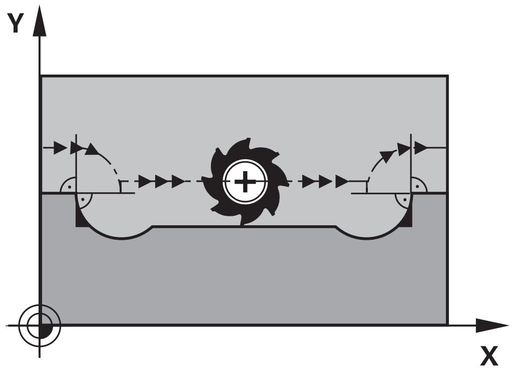
7.4 Miscellaneous functions for path behavior

Machining small contour steps: M97

Standard behavior

The control inserts a transition arc at outside corners. For very small contour steps, the tool would damage the contour.

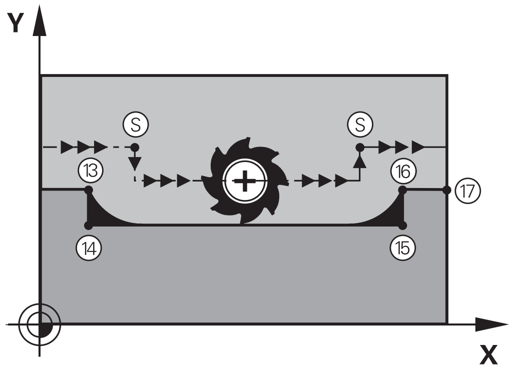
In such cases, the control interrupts the program run and generates the **Tool radius too large** error message.



Behavior with M97

The control determines a path intersection for the contour elements—such as inner corners—and moves the tool above this point.

Program **M97** in the same NC block as the outside corner.



> HEIDENHAIN recommends to use the much more powerful **M120 LA** function instead of **M97** here.
> **Further information:** "Pre-calculating radius-compensated contours (LOOK AHEAD): M120 ", Page 233

Effect

M97 is effective only in the NC block in which **M97** is programmed.

> The control does not completely finish the corner when it is machined with **M97**. You may wish to rework the contour with a smaller tool.

Example

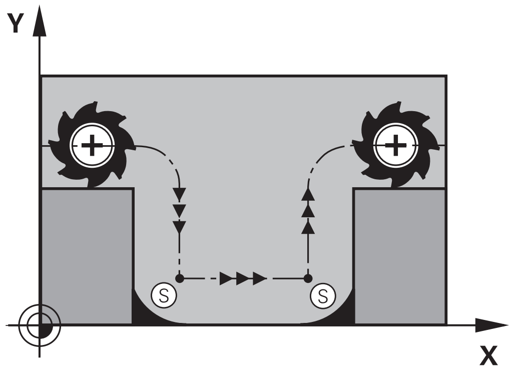
5 TOOL DEF L ... R+20	Large tool radius
...	
13 L X... Y... R... F... M97	Move to contour point 13
14 L IY-0.5 ... R... F...	Machine small contour step 13 to 14
15 L IX+100 ...	Move to contour point 15
16 L IY+0.5 ... R... F... M97	Machine small contour step 15 to 16
17 L X... Y...	Move to contour point 17

Machining open contour corners: M98

Standard behavior

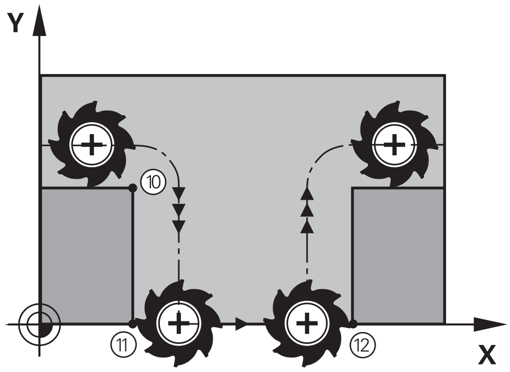
The control calculates the intersections of the cutter paths at inside corners and moves the tool in the new direction at those points.

If the contour is open at the corners, however, this will result in incomplete machining.



Behavior with M98

With the **M98** miscellaneous function, the control temporarily suspends radius compensation to ensure that both corners are completely machined:



Effect

M98 is effective only in the NC blocks in which **M98** is programmed.

M98 becomes effective at the end of the block.

Example: Move to the contour points 10, 11 and 12 in succession

```
10 L X... Y...  RL F
11 L X... IY...  M98
12  L IX+ ...
```

Feed rate factor for plunging movements: M103

Standard behavior

The control moves the tool at the last programmed feed rate, regardless of the direction of traverse.

Behavior with M103

The control reduces the feed rate when the tool moves in the negative direction of the tool axis. The feed rate for plunging FZMAX is calculated from the last programmed feed rate FPROG and a factor F%:

FZMAX = FPROG x F%

Programming M103

If you program **M103** in a positioning block, the control continues the dialog by prompting you for the F factor.

Effect

M103 becomes effective at the start of the block.
Cancel **M103**: Program **M103** once again without a factor.

> The **M103** is also effective with an active tilted working plane coordinate system. The feed rate reduction is then effective in the negative direction when moving the **tilted** tool axis.

Example

The feed rate for plunging is to be 20% of the feed rate in the plane.

...	Actual contouring feed rate (mm/min):
17 L X+20 Y+20 RL F500 M103 F20	500
18 L Y+50	500
19 L IZ-2.5	100
20 L IY+5 IZ-5	141
21 L IX+50	500
22 L Z+5	500

Feed rate in millimeters per spindle revolution: M136

Standard behavior

The control moves the tool at the feed rate F in mm/min programmed in the NC program

Behavior with M136

> In NC programs based on inch units, **M136** is not allowed in combination with the alternative **FU** feed rate.
>
> The spindle is not permitted to be controlled when M136 is active.

With **M136**, the control does not move the tool in mm/min, but rather at the feed rate F in millimeters per spindle revolution programmed in the NC program. If you change the spindle speed by using the potentiometer, the control changes the feed rate accordingly.

Effect

M136 becomes effective at the start of the block.

You can cancel **M136** by programming **M137**.

Feed rate for circular arcs: M109/M110/M111

Standard behavior

The control applies the programmed feed rate to the path of the tool center.

Behavior at circular arcs with M109

For inside and outside machining of circular arcs, the control keeps the feed rate at the cutting edge constant.

NOTICE

Caution: Danger to the tool and workpiece!

If the **M109** function is active, the control might dramatically increase the feed rate when machining very small outside corners. During the execution, there is a risk of tool breakage or workpiece damage.

- Do not use **M109** for machining very small outside corners

Behavior at circular arcs with M110

With circular arcs, the control only keeps the feed rate constant for inside machining operations. The feed rate will not be adjusted for outside machining of circular arcs.

> If you program **M109** or **M110** with a number > 200 before calling a machining cycle, the adjusted feed rate will also be effective for circular arcs within these machining cycles. The initial state is restored after finishing or canceling a machining cycle.

Effect

M109 and **M110** become effective at the start of the block. **M109** and **M110** can be canceled with **M111**.

Pre-calculating radius-compensated contours (LOOK AHEAD): M120

Standard behavior

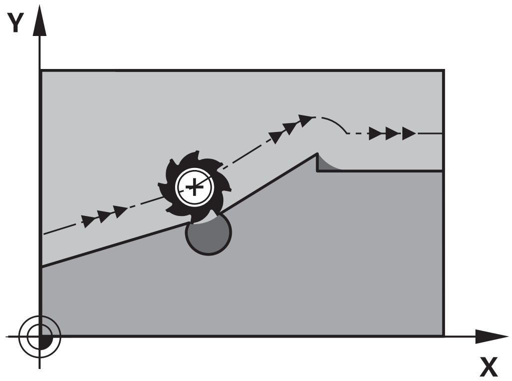
If the tool radius is larger than the contour step that needs to be machined with radius compensation, the control interrupts program run and generates an error message. **M97** inhibits the error message, but this results in dwell marks and will also move the corner.

Further information: "Machining small contour steps: M97", Page 229

The control might damage the contour in case of undercuts.

Behavior with M120

The control checks radius-compensated contours for undercuts and tool path intersections, and calculates the tool path in advance from the current NC block. Areas of the contour that would be damaged by the tool will not be machined (shown darker in the figure). You can also use **M120** to calculate the tool radius compensation for digitized data or data created on an external programming system. This means that deviations from the theoretical tool radius can be compensated.



The number of NC blocks (99 max.) that are calculated in advance can be defined with **LA** (**L**ook **A**head) following **M120**. Note that the larger the number of NC blocks you choose, the higher the block processing time will be.

Input

If you enter **M120** in a positioning block, the control continues the dialog for this NC block by prompting you for the number of **LA** NC blocks to be calculated in advance.

Effect

Program the **M120** function in an NC block that also contains an **RL** or **RR** radius compensation. This way, you can achieve consistent programming, resulting in clearly structured programs. You can deactivate the **M120** function with the following NC syntax:

- **R0**
- **M120 LA0**
- **M120** without **LA**
- **PGM CALL**
- Cycle **19** or **PLANE** functions

M120 becomes effective at the start of the block.

Restrictions

- After an external or internal stop, you can only re-enter the contour with the function **RESTORE POS. AT N**. Before you start the block scan, you must cancel **M120**, otherwise the control will generate an error message.
- If you want to approach the contour on a tangential path, you must use the **APPR LCT** function. The NC block with **APPR LCT** must contain only the coordinates of the working plane.
- If you want to depart the contour on a tangential path, you must use the function **DEP LCT**. The NC block with **DEP LCT** must contain only the coordinates of the working plane.
- Before using the functions listed below, you have to cancel **M120** and the radius compensation:
  - Cycle **32** Tolerance
  - Cycle **19** Working plane
  - **PLANE** function
  - **M114**
  - **M128**
  - **TCPM FUNCTION**

Superimposing handwheel positioning during program run: M118

Standard behavior

> Refer to your machine manual!
> Your machine tool builder must have prepared the control for this function.

In the Program Run operating modes, the control moves the tool as defined in the NC program.

Behavior with M118

M118 permits manual corrections by handwheel during the program run. For this purpose, you program **M118** and enter an axis-specific value (linear or rotary axis).

> The **M118** handwheel superimpositioning function, in combination with the **Dynamic Collision Monitoring (DCM)** function, can only be used at a standstill.
> In order to use **M118** without restrictions, either deselect the **Dynamic Collision Monitoring (DCM)** function using the soft key from the menu or activate a kinematics model without collision objects (CMOs).

Input

If you enter **M118** in a positioning block, the control continues the dialog for this block by prompting you for the axis-specific values. Use the orange axis keys or the alphabetic keyboard for entering the coordinates.

Effect

To cancel handwheel positioning, program **M118** once again without coordinate input or end the NC program with **M30** / **M2**.

> If the program aborts, handwheel positioning will also be canceled.

M118 becomes effective at the start of the block.

Example

You want to be able to use the handwheel during program run to move the tool in the working plane X/Y by ±1 mm and in the rotary axis B by ±5° from the programmed value:

```
L X+0  Y+38.5 RL  F125 M118  X1 Y1 B5
```

When programmed in an NC program, **M118** is always effective in the machine coordinate system.

If the Global Program Settings option (Option 44) is active, **Handradüberlagerung** is effective in the last selected coordinate system. The coordinate system active for Handradüberlagerung is shown on the **POS HR** tab of the additional status display.

The **POS HR** tab also indicates whether the **Max. val.** have been defined via **M118** or via the Global Program Settings.

Further information: User's Manual for Setup, Testing and Running NC Programs

Handradüberlagerung is also effective in the **Positioning w/ Manual Data Input** operating mode!

Virtual tool axis (VT)(Option 44)

Refer to your machine manual!

Your machine tool builder must have prepared the control for this function.

With the virtual tool axis, you can also traverse with the handwheel in the direction of a sloping tool on a machine with swivel heads. To traverse in a virtual tool axis direction, select the **VT** axis on the display of your handwheel.

Further information: User's Manual for Setup, Testing and Running NC Programs

When using an HR 5xx handwheel, you can select the virtual axis directly with the orange **VI** axis key, if necessary.

In conjunction with the **M118** function, it is also possible to carry out handwheel superimpositioning in the currently active tool axis direction. For this purpose, program at least the spindle axis with its permitted range of traverse in the **M118** function (e.g. **M118 Z5**) and select the **VT** axis on the handwheel.

Retraction from the contour in the tool-axis direction: M140

Standard behavior

In the **Program Run Single Block** and **Program Run Full Sequence** operating modes, the control moves the tool as defined in the NC program.

Behavior with M140

With **M140 MB** (move back), you can retract the tool from the contour by a programmable distance in the direction of the tool axis.

NOTICE

Danger of collision!

The machine tool builder has various options for configuring the **Dynamic Collision Monitoring (DCM)** function. Depending on the machine, the NC program will be continued without an error message despite a detected collision, but the tool will be stopped at the last position without collision. If the NC program allows for a new position without collision, the control resumes the machining operation and positions the tool at that position. This configuration of the **Dynamic Collision Monitoring (DCM)** function results in movements that are not defined in the program. **This process takes place no matter whether collision monitoring is active or inactive.** There is a danger of collision during these movements!

- Refer to your machine manual.
- Check the behavior at the machine.

Input

If you enter **M140** in a positioning block, the control continues the dialog and prompts you for the path the tool should use for retracting from the contour. Enter the desired path that the tool should follow when retracting from the contour, or press the **MB MAX** soft key to move to the limit of the traverse range.

In the optional machine parameter **moveBack** (no. 200903), the machine tool builder defines how far before a limit switch or a collision object a retraction movement **MB MAX** should end.

In addition, you can program the feed rate at which the tool traverses the entered path. If you do not enter a feed rate, the control moves the tool along the entered path at rapid traverse.

Effect

M140 is effective only in the NC block in which itis programmed.

M140 becomes effective at the start of the block.

Example

NC block 250: Retract the tool by 50 mm from the contour

NC block 251: Move the tool to the limit of the traverse range

```
250 L  X+0 Y+38.5  F125 M140  MB 50 F750
251 L  X+0 Y+38.5  F125 M140  MB MAX
```

M140 is also effective if the **Tilt working plane** function is active. For machines with swivel heads the control then moves the tool in the tilted coordinate system.

With **M140 MB MAX** you can only retract in the positive direction.

Always define a tool call with tool axis before **M140**, otherwise the traverse direction is not defined.

NOTICE

Danger of collision!

If you use the **M118** function to modify the position of a rotary axis with the handwheel and then execute the **M140** function, the control ignores the superimposed values with the retraction movement. This results in unwanted and unpredictable movements, especially when using machines with head rotation axes. There is a danger of collision during these compensating movements!

- Do not combine **M118** with **M140** when using machines with head rotation axes.

Suppressing touch probe monitoring: M141

Standard behavior

If the stylus is deflected, the control issues an error message as soon as you want to move a machine axis.

Behavior with M141

The control moves the machine axes even if the touch probe is deflected. This function is required if you wish to write your own measuring cycle in connection with measuring cycle 3 in order to retract the stylus by means of a positioning block after it has been deflected.

NOTICE

Danger of collision!

The function **M141** suppresses the corresponding error message if the stylus is deflected. 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 test the NC program or program section in **Program run, single block** operating mode

M141 functions only for movements with straight-line blocks.

Effect

M141 is effective only in the NC block in which **M141** is programmed.

M141 becomes effective at the start of the block.

Deleting basic rotation: M143

Standard behavior

The basic rotation remains in effect until it is reset or is overwritten with a new value.

Behavior with M143

The control deletes a basic rotation from the NC program.

> The function **M143** is not permitted with mid-program startup.

Effect

M143 is effective only from the NC block in which it is programmed.

M143 becomes effective at the start of the block.

> **M143** clears the entries from the **SPA**, **SPB** and **SPC** columns in the preset table. When the corresponding line is reactivated, the basic rotation is **0** in all columns.

Automatically retracting the tool from the contour at an NC stop: M148

Standard behavior

In case of an NC stop, the control stops all traverse movements. The tool stops moving at the point of interruption.

Behavior with M148

> Refer to your machine manual!
>
> This function must be configured and enabled by your machine tool builder.
>
> In the **CfgLiftOff** (no. 201400) machine parameter, the machine tool builder defines the path the control is to traverse for a **LIFTOFF** command. You can also use the **CfgLiftOff** machine parameter to deactivate the function.

Set the **Y** parameter in the **LIFTOFF** column of the tool table for the active tool. The control then retracts the tool from the contour by 2 mm max. in the direction of the tool axis.

Further information: User's Manual for Setup, Testing and Running NC Programs

LIFTOFF takes effect in the following situations:

- An NC stop triggered by you
- An NC stop triggered by the software, e.g. if an error occurred in the drive system
- When a power interruption occurs

Effect

M148 remains in effect until deactivated with **M149**.

M148 becomes effective at the start of the block, **M149** at the end of the block.

Rounding corners: M197

Standard behavior

With active radius compensation, the control inserts a transition arc at outside corners. This may lead to rounding of that edge.

Behavior with M197

With the **M197** function, the contour at the corner is tangentially extended and a smaller transition arc is then inserted. When you program the **M197** function and then press the **ENT** key, the control opens the **DL** input field. In **DL**, you define the length the control by which the control extends the contour elements. With **M197**, the corner radius is reduced, the corner is rounded less and the traverse movement is still smooth.

Effect

The **M197** function acts blockwise and is only effective on outside corners.

Example

```
L  X... Y...  RL M197 DL0.876
```

8

Subprograms and Program Section Repeats

8.1 Labeling subprograms and program section repeats

Subprograms and program section repeats enable you to program a machining sequence once and then run it as often as necessary.

Label

The beginnings of subprograms and program section repeats in NC programs are marked by **(LBL)** labels.

A LABEL contains a number between 1 and 65535 or a name definable by you. Each LABEL number or LABEL name can be used only once within the NC program and is set with the **LABEL SET** key. The quantity of label names you are able to enter is limited only by the amount of internal memory.

> Do not use a label number or label name more than once!

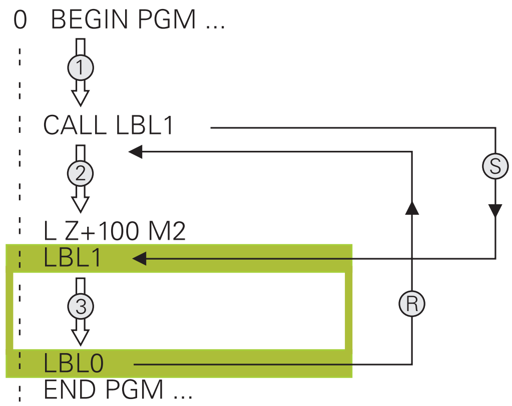
Label 0 (**LBL 0**) is used exclusively to mark the end of a subprogram and can therefore be used as often as desired.

> Before creating your NC program, compare the subprogram and program section repeat programming techniques using the so-called If-Then Decisions.
>
> You can thereby avoid possible misunderstandings and programming errors.
>
> **Further information:** "If-then decisions with Q parameters", Page 275

8.2 Subprograms

Operating sequence

1. The control executes the NC program up to the block in which a subprogram is called with **CALL LBL**
2. The subprogram is then executed until the subprogram end **LBL 0**
3. The control then resumes the NC program from the NC block after the subprogram call **CALL LBL**



Programming notes

- A main program can contain any number of subprograms
- You can call subprograms in any sequence and as often as desired
- A subprogram cannot call itself
- Write subprograms after the NC block with M2 or M30
- If subprograms are located in the NC program before the NC block with M2 or M30, they will be executed at least once even if they are not called

Programming the subprogram

LBL SET

- To mark the beginning: Press the **LBL SET** key
- Enter the subprogram number. If you want to use a label name, press the **LBL NAME** soft key to switch to text entry.
- Enter the text
- Mark the end: Press the **LBL SET** key and enter the label number **0**

Calling a subprogram

LBL CALL

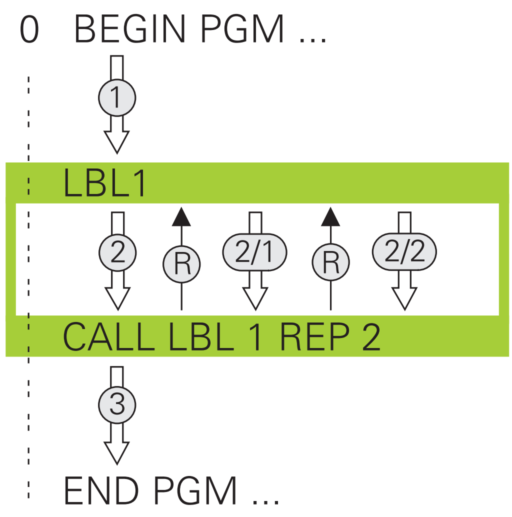
- Call a subprogram: Press the **LBL CALL** key
- Enter the subprogram number of the subprogram you wish to call. If you want to use a label name, press the **LBL NAME** soft key to switch to text entry.
- If you want to enter the number of a string parameter as target address, press the QS soft key
- > The control then jumps to the label name that is specified in the string parameter defined.
- Ignore repeats **REP** by pressing the **NO ENT** key. Repeat **REP** is used only for program section repeats

CALL LBL 0 is not permitted (Label 0 is only used to mark the end of a subprogram).

8.3 Program-section repeats

Label

The beginning of a program section repeat is marked by the label **LBL**. The end of a program section repeat is identified by **CALL LBL n REPn**.



Operating sequence

1. The control executes the NC program up to the end of the program section (**CALL LBL n REPn**)
2. Then the program section between the called LABEL and the label call **CALL LBL n REPn** is repeated the number of times entered after **REP**
3. The control then resumes the NC program after the last repetition.

Programming notes

- You can repeat a program section up to 65 534 times in succession
- The total number of times the program section is executed is always one more than the programmed number of repeats, because the first repeat starts after the first machining process.

Programming a program section repeat

LBL SET

- To mark the beginning, press the **LBL SET** key and enter a LABEL NUMBER for the program section you wish to repeat. If you want to use a label name, press the **LBL NAME** soft key to switch to text entry.
- Enter the program section

Calling a program section repeat

LBL CALL

- Call a program section: Press the **LBL CALL** key
- Enter the program section number of the program section to be repeated. If you want to use a LABEL name, press the **LBL NAME** soft key to switch to text entry
- Enter the number of repeats **REP** and confirm with the **ENT** key.

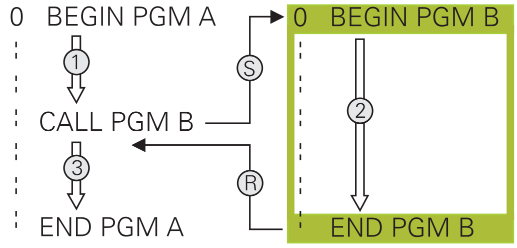
8.4 Calling an external NC program

Overview of the soft keys

When you press the **PGM CALL** key, the control displays the following soft keys:

Soft key	Function
CALL PROGRAM	Call an NC program with **PGM CALL**
SELECT DATUM TABLE	Select a datum table with **SEL TABLE**
SELECT POINT TABLE	Select a point table with **SEL PATTERN**
SELECT CONTOUR	Select a contour program with **SEL CONTOUR**
SELECT PROGRAM	Select an NC program with **SEL PGM**
CALL SELECTED PROGRAM	Call the last selected file with **CALL SELECTED PGM**
SELECT CYCLE	Select any NC program with **SEL CYCLE** as a fixed cycle **Further information:** Cycle Programming User's Manual

Operating sequence



1. The control executes the NC program up to the block in which another NC program is called with **CALL PGM**.
2. Then the other NC program is run from beginning to end.
3. The control then resumes the calling NC program with the NC block behind the program call.

> If you want to program variable program calls in connection with string parameters, use the **SEL PGM** function.

Programming notes

- The control does not require any labels to call an NC program.
- The called NC program must not have a **CALL PGM** call into the calling NC program (an endless loop ensues).
- The called NC program must not contain the miscellaneous function **M2** or **M30**. If you have defined subprograms with labels in the called NC program, then you can replace M2 or M30 with the jump function **FN 9: If +0 EQU +0 GOTO LBL 99**.
- If you want to call a ISO program, enter the file type .I after the program name.
- You can also call an NC program with Cycle **12 PGM CALL**.
- You can also call any NC program with the function **Select the cycle** (**SEL CYCLE**).
- As a rule, Q parameters are effective globally with a **PGM CALL**. So please note that changes to Q parameters in the called NC program can also influence the calling NC program.

> While the control is running the calling NC program, the editing of all called NC programs is disabled.

Checking the called NC programs

NOTICE

Danger of collision!

The control does not automatically check whether collisions can occur between the tool and the workpiece. If you do not specifically rescind the coordinate transformations in the called NC program, these transformations will also take effect in the calling NC program. Danger of collision during machining!

- Reset used coordinate transformations in the same NC program
- Check the machining sequence using a graphic simulation if required

The control checks the called NC programs:

- If the called NC program contains the miscellaneous functions **M2** or **M30**, then the control displays a warning. The control automatically clears the warning as soon as you select another NC program.
- The control checks the called NC programs to see whether they are complete before running them. If the **END PGM** NC block is missing, the control aborts with an error message.

Further information: User's Manual for Setup, Testing and Running NC Programs

Path information

If the NC program you want to call is located in the same directory as the NC program you are calling it from, then you only need to enter the program name.

If the called NC program is not located in the same directory as the NC program you are calling it from, you must enter the complete path, e.g. **TNC:\ZW35\HERE\PGM1.H**

Alternatively, you can program relative paths:

- Starting from the folder of the calling NC program one folder level up **..\PGM1.H**
- Starting from the folder of the calling NC program one folder level down **DOWN\PGM1.H**
- Starting from the folder of the calling NC program one folder level up and in one other folder **..\THERE\PGM3.H**

Calling an external NC program

Calling a program with PGM CALL

You can call an external NC program with the **PGM CALL** function. The control runs the external NC program from the position where it was called in the NC program.

Proceed as follows:

PGM CALL

- Press the **PGM CALL** key

CALL PROGRAM

- Press the **CALL PROGRAM** soft key
- > The control starts the dialog for defining the NC program to be called.
- Enter the path name with the keyboard

Alternative:

SELECT FILE

- Press the **SELECT FILE** soft key
- > The control displays a selection window in which you can select the NC program to be called.
- Press the **ENT** key

Call with SEL PGM and CALL SELECTED PGM

The function **SEL PGM** Allows you to select an external NC program that you can separately call at a different position in the NC program. The control runs the external NC program from the position at which you called it in the NC program using **CALL SELECTED PGM**.

The **SEL PGM** function is also permitted with string parameters, so that you can dynamically control program calls.

To select the NC program, proceed as follows:

PGM CALL

- Press the **PGM CALL** key

SELECT PROGRAM

- Press the **SELECT PROGRAM** soft key
- > The control starts the dialog for defining the NC program to be called.

SELECT FILE

- Press the **SELECT FILE** soft key
- > The control displays a selection window in which you can select the NC program to be called.
- Press the **ENT** key

To call the selected NC program, proceed as follows:

PGM CALL

- Press the **PGM CALL** key

CALL SELECTED PROGRAM

- Press the **CALL SELECTED PROGRAM** soft key
- > The control uses **CALL SELECTED PGM** to call the NC program that was selected last.

> If an NC program that was called using **CALL SELECTED PGM** is missing, then the control interrupts the execution or simulation with an error message. In order to avoid undesired interruptions during program run, you can use the function **FN 18** (**ID10 NR110** and **NR111**) to check all paths at the beginning of the program.
> **Further information:** "FN 18: SYSREAD – Reading system data", Page 293

8.5 Nesting

Types of nesting

- Subprogram calls in subprograms
- Program-section repeats within a program-section repeat
- Subprogram calls in program section repeats
- Program-section repeats in subprograms

> Subprograms and program section repeats can call external NC programs as well.

Nesting depth

The nesting depth defines, among other things, how often program sections or subprograms may contain further subprograms or program section repeats.

- Maximum nesting depth for subprograms: 19
- Maximum nesting depth for external NC programs: 19, for which a **CYCL CALL** has the effect of calling an external program
- You can nest program section repeats as often as desired

Subprogram within a subprogram

Example

0 BEGIN PGM UPGMS MM	
...	
17 CALL LBL "UP1"	Call the subprogram marked with LBL SP1
...	
35 L Z+100 R0 FMAX M2	Last program block of the main program with M2
36 LBL "UP1"	Beginning of subprogram SP1
...	
39 CALL LBL 2	Call the subprogram marked with LBL 2
...	
45 LBL 0	End of subprogram 1
46 LBL 2	Beginning of subprogram 2
...	
62 LBL 0	End of subprogram 2
63 END PGM SUBPGMS MM	

Program execution

1. Main program UPGMS is executed up to NC block 17
2. Subprogram UP1 is called, and executed up to NC block 39
3. Subprogram 2 is called, and executed up to NC block 62. End of subprogram 2 and return jump to the subprogram from which it was called.
4. Subprogram UP1 is called, and executed from NC block 40 up to NC block 45. End of subprogram 1 and return jump to the main program UPGMS.
5. Main program UPGMS is executed from NC block 18 up to NC block 35. Return jump to NC block 1 and end of program

Repeating program section repeats

Example

0 BEGIN PGM REPS MM	
...	
15 LBL 1	Beginning of program section repeat 1
...	
20 LBL 2	Beginning of program section repeat 2
...	
27 CALL LBL 2 REP 2	Program section call with two repeats
...	
35 CALL LBL 1 REP 1	The program section between this NC block and LBL 1
...	(NC block 15) is repeated once
50 END PGM REPS MM	

Program execution

1. Main program REPS is executed up to NC block 27
2. The program section between NC block 27 and NC block 20 is repeated twice
3. Main program REPS is executed from NC block 28 up to NC block 35
4. The program section between NC block 35 and NC block 15 is repeated once (including the program section repeat between NC block 20 and NC block 27)
5. Main program REPS is executed from NC block 36 up to NC block 50. Return jump to NC block 1 and end of program

Repeating a subprogram

Example

0 BEGIN PGM UPGREP MM	
...	
10 LBL 1	Beginning of program section repeat 1
11 CALL LBL 2	Subprogram call
12 CALL LBL 1 REP 2	Program section call with two repeats
...	
19 L Z+100 R0 FMAX M2	Last NC block of the main program with M2
20 LBL 2	Beginning of subprogram
...	
28 LBL 0	End of subprogram
29 END PGM UPGREP MM	

Program execution

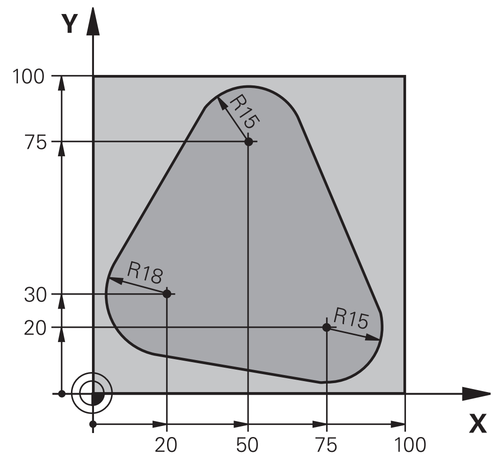
1 Main program UPGREP is executed up to NC block 11
2 Subprogram 2 is called and executed.
3 The program section between NC block 12 and NC block 10 is repeated twice. This means that subprogram 2 is repeated twice
4 Main program UPGREP is executed from NC block 13 up to NC block 19. Return jump to NC block 1 and end of program

8.6 Programming examples

Example: Milling a contour in several infeeds

Program run:

- Pre-position the tool to the workpiece surface
- Enter the infeed depth in incremental values
- Contour milling
- Repeat infeed and contour-milling

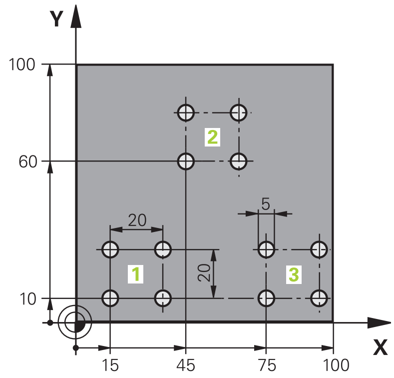


0 BEGIN PGM PGMWDH MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-40	
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 1 Z S500	Tool call
4 L Z+250 R0 FMAX	Retract the tool
5 L X-20 Y+30 R0 FMAX	Pre-position in the working plane
6 L Z+0 R0 FMAX M3	Pre-position to the workpiece surface
7 LBL 1	Set label for program section repeat
8 L IZ-4 R0 FMAX	Incremental downfeed (in the air)
9 APPR CT X+2 Y+30 CCA90 R+5 RL F250	Contour approach
10 FC DR- R18 CLSD+ CCX+20 CCY+30	Contour
11 FLT	
12 FCT DR- R15 CCX+50 CCY+75	
13 FLT	
14 FCT DR- R15 CCX+75 CCY+20	
15 FLT	
16 FCT DR- R18 CLSD- CCX+20 CCY+30	
17 DEP CT CCA90 R+5 F1000	Contour departure
18 L X-20 Y+0 R0 FMAX	Retract tool
19 CALL LBL 1 REP 4	Return jump to LBL 1; section is repeated a total of 4 times
20 L Z+250 R0 FMAX M2	Retract the tool, end of program
21 END PGM PGMWDH MM	

Example: Groups of holes

Program run:

- Approach the groups of holes in the main program
- Call the group of holes (subprogram 1) in the main program
- Program the group of holes only once in subprogram 1

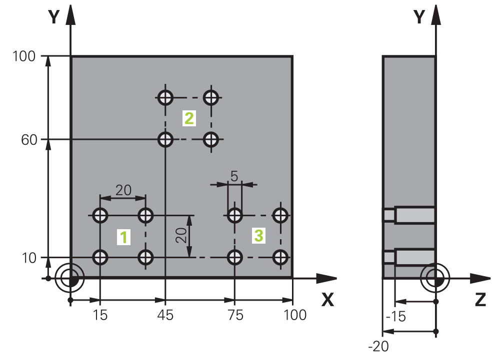


0 BEGIN PGM UP1 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 1 Z S5000	Tool call
4 L Z+250 R0 FMAX	Retract the tool
5 CYCL DEF 200 DRILLING	Cycle definition: Drilling
Q200=2 ;SET-UP CLEARANCE	
Q201=-10 ;DEPTH	
Q206=250 ;FEED RATE FOR PLNGNG	
Q202=5 ;PLUNGING DEPTH	
Q210=0 ;DWELL TIME AT TOP	
Q203=+0 ;SURFACE COORDINATE	
Q204=10 ;2ND SET-UP CLEARANCE	
Q211=0.25 ;DWELL TIME AT DEPTH	
Q395=0 ;DEPTH REFERENCE	
6 L X+15 Y+10 R0 FMAX M3	Move to starting point for group 1
7 CALL LBL 1	Call the subprogram for the group
8 L X+45 Y+60 R0 FMAX	Move to starting point for group 2
9 CALL LBL 1	Call the subprogram for the group
10 L X+75 Y+10 R0 FMAX	Move to starting point for group 3
11 CALL LBL 1	Call the subprogram for the group
12 L Z+250 R0 FMAX M2	End of main program
13 LBL 1	Beginning of subprogram 1: Group of holes
14 CYCL CALL	Hole 1
15 L IX+20 R0 FMAX M99	Move to 2nd hole, call cycle
16 L IY+20 R0 FMAX M99	Move to 3rd hole, call cycle
17 L IX-20 R0 FMAX M99	Move to 4th hole, call cycle
18 LBL 0	End of subprogram 1
19 END PGM UP1 MM	

Example: Group of holes with multiple tools

Program run:

- Program the fixed cycles in the main program
- Call the complete hole pattern (subprogram 1) in the main program
- Approach the groups of holes (subprogram 2) in subprogram 1
- Program the group of holes only once in subprogram 2



0 BEGIN PGM UP2 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 1 Z S5000	Centering drill tool call
4 L Z+250 R0 FMAX	Retract the tool
5 CYCL DEF 200 DRILLING	Cycle definition: Centering
Q200=2 ;SET-UP CLEARANCE	
Q201=-3 ;DEPTH	
Q206=250 ;FEED RATE FOR PLNGNG.	
Q202=3 ;PLUNGING DEPTH	
Q210=0 ;DWELL TIME AT TOP	
Q203=+0 ;SURFACE COORDINATE	
Q204=10 ;2ND SET-UP CLEARANCE	
Q211=0.25 ;DWELL TIME AT DEPTH	
Q395=0 ;DEPTH REFERENCE	
6 CALL LBL 1	Call subprogram 1 for the entire hole pattern
7 L Z+250 R0 FMAX	
8 TOOL CALL 2 Z S4000	Drill tool call
9 FN 0: Q201 = -25	New depth for drilling
10 FN 0: Q202 = +5	New plunging depth for drilling
11 CALL LBL 1	Call subprogram 1 for the entire hole pattern
12 L Z+250 R0 FMAX	
13 TOOL CALL 3 Z S500	Reamer tool call

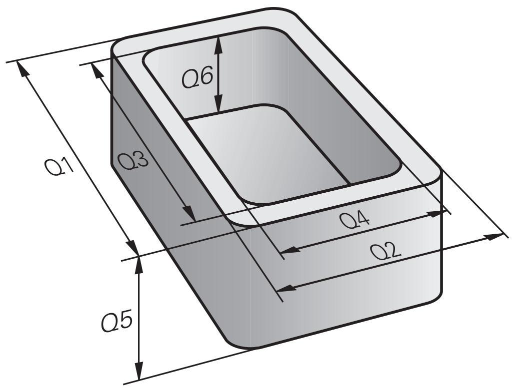
14 CYCL DEF 201 REAMING	Cycle definition: Reaming
Q200=2 ;SET-UP CLEARANCE	
Q201=-15 ;DEPTH	
Q206=250 ;FEED RATE FOR PLNGNG.	
Q211=0.5 ;DWELL TIME AT DEPTH	
Q208=400 ;RETRACTION FEED RATE	
Q203=+0 ;SURFACE COORDINATE	
Q204=10 ;2ND SET-UP CLEARANCE	
15 CALL LBL 1	Call subprogram 1 for the entire hole pattern
16 L Z+250 R0 FMAX M2	End of main program
17 LBL 1	Beginning of subprogram 1: Entire hole pattern
18 L X+15 Y+10 R0 FMAX M3	Move to starting point for hole group 1
19 CALL LBL 2	Call subprogram 2 for the hole group
20 L X+45 Y+60 R0 FMAX	Move to starting point for hole group 2
21 CALL LBL 2	Call subprogram 2 for the hole group
22 L X+75 Y+10 R0 FMAX	Move to starting point for hole group 3
23 CALL LBL 2	Call subprogram 2 for the hole group
24 LBL 0	End of subprogram 1
25 LBL 2	Beginning of subprogram 2: Group of holes
26 CYCL CALL	Hole 1 with active machining cycle
27 L IX+20 R0 FMAX M99	Move to 2nd hole, call cycle
28 L IY+20 R0 FMAX M99	Move to 3rd hole, call cycle
29 L IX-20 R0 FMAX M99	Move to 4th hole, call cycle
30 LBL 0	End of subprogram 2
31 END PGM SP2 MM	

9

Programming Q Parameters

9.1 Principle and overview of functions

With Q parameters you can program entire families of parts in a single NC program by programming variable Q parameters instead of fixed numerical values.



Q parameters can be used in the following ways:

- Coordinate values
- Feed rates
- Spindle speeds
- Cycle data

The control provides more ways to use Q parameters:

- Program contours that are defined through mathematical functions
- Make execution of machining steps depend on certain logical conditions
- Variably design FK programs

Q parameters are always identified with letters and numbers. The letters determine the type of Q parameter and the numbers the Q parameter range.

For more information, see the table below:

Q parameter type	Q parameter range	Meaning
Q parameters:		**Parameters affect all NC programs in the control's memory**
	0 to 99	Parameters for the **user**, if there are no overlaps with the HEIDENHAIN-SL cycles These parameters have a local effect within so-called macros and OEM cycles. This means that changes are not returned to the NC program. For this reason, use the Q parameter range 1200 – 1399 for OEM cycles!
	100 to 199	Parameters for special functions on the control that can be read by NC programs of the user or by cycles
	200 to 1199	Parameters primarily used for HEIDENHAIN cycles
	1200 to 1399	Parameters preferentially used with manufacturer cycles if values are returned to the user program
	1400 to 1599	Parameters primarily used as input parameters for manufacturer cycles
	1600 to 1999	Parameters for **users**
QL parameters:		**Parameters only effective locally within an NC program**
	0 to 499	Parameters for **users**
QR parameters:		**Parameters permanently affect all NC programs in the control's memory, including after a power interruption**
	0 to 99	Parameters for **users**
	100 to 199	Parameters for HEIDENHAIN functions (e.g., cycles)
	200 to 499	Parameters for the machine tool builder (e.g., cycles)

QR parameters will be included in backups.

If the machine tool builder did not define a specific path, the control will save the **QR** parameter values in the following path: **SYS:\runtime\sys.cfg**. This partition will only be backed up in full backups.

Machine tool builders can use the following optional machine parameters to specify the paths:

- **pathNcQR** (no. 131201)
- **pathSimQR** (no. 131202)

If the machine tool builder used the optional machine parameters to specify a path on the TNC partition, you can perform a backup with the **NC/PLC Backup** functions without entering a code number.

QS parameters (**S** stands for string) are also available and enable you to process texts on the control.

Q parameter type	Q parameter range	Meaning
QS parameters:		**Parameters affect all NC programs in the control's** memory
	0 to 99	Parameters for the **user**, where no overlaps with the HEIDENHAIN SL cycles are present These parameters have a local effect within so-called macros and OEM cycles. This means that changes are not returned to the NC program. For this reason, use the QS parameter range 200 – 499 for OEM cycles!
	100 to 199	Parameters for special functions on the control that can be read by NC programs of the user or by cycles
	200 to 1199	Parameters primarily used for HEIDENHAIN cycles
	1200 to 1399	Parameters preferentially used with manufacturer cycles if values are returned to the user program
	1400 to 1599	Parameters primarily used as input parameters for manufacturer cycles
	1600 to 1999	Parameters for **users**

NOTICE

Danger of collision!

HEIDENHAIN cycles, manufacturer cycles and third-party functions use Q parameters. You can also program Q parameters within NC programs. If, when using Q parameters, the recommended Q parameter ranges are not used exclusively, then this can lead to overlapping (reciprocal effects) and thus cause undesired behavior. Danger of collision during machining!

- Only use Q parameter ranges recommended by HEIDENHAIN.
- Comply with the documentation from HEIDENHAIN, the machine tool builder, and suppliers.
- Check the machining sequence using a graphic simulation

Programming notes

You can mix Q parameters and numerical values within an NC program.

Q parameters can be assigned numerical values between -999 999 999 and +999 999 999. The input range is limited to 16 digits, of which 9 may be before the decimal point. Internally the control calculates numbers up to a value of 10^{10}.

You can assign a maximum of 255 characters to **QS** parameters.

> The control automatically assigns some Q and QS parameters the same data, e.g., the Q parameter **Q108** is automatically assigned the current tool radius.
>
> **Further information:** "Preassigned Q parameters", Page 337
>
> The control saves numerical values internally in a binary number format (standard IEEE 754). Due to the standardized format used, the control does not represent some decimal numbers with a binary number that is 100% exact (round-off error). If you use calculated Q parameter contents for jump commands or positioning moves, then you must take this fact into consideration.

You can reset Q parameters to the status **Undefined**. If a position is programmed with a Q parameter that is undefined, the control ignores this movement.

Calling Q parameter functions

When you are writing an NC program, press the **Q** key (in the numeric keypad for numerical input and axis selection, below the **+/-** key). The control then displays the following soft keys:

Soft key	Function group	Page
BASIC ARITHM.	Basic arithmetic (assign, add, subtract, multiply, divide, square root)	270
TRIGO-NOMETRY	Trigonometric functions	273
CIRCLE CALCU-LATION	Function for calculating circles	274
JUMP	If/then conditions, jumps	275
DIVERSE FUNCTION	Other functions	280
FORMULA	Entering formulas directly	320
CONTOUR FORMULA	Function for machining complex contours	See Cycle Programming User's Manual

If you define or assign a Q parameter, then the control shows the **Q**, **QL** and **QR** soft keys. You can use these soft keys to select the desired parameter type. Then you define the parameter number.

9.2 Part families—Q parameters in place of numerical values

Application

The Q parameter function **FN 0: Assign** allows you to assign numerical values to Q parameters. You then use a Q parameter in place of the numerical value in the NC program.

Example

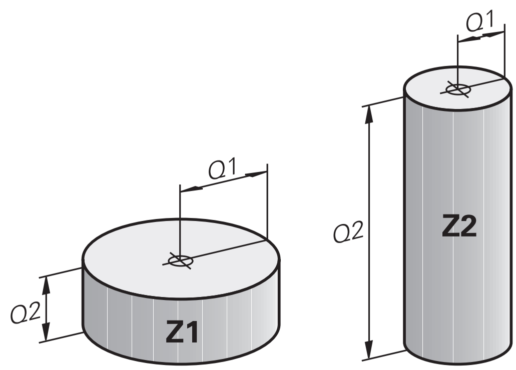
15 FN 0: Q10=25	Assign
...	Q10 is assigned the value 25
25 L X +Q10	Means L X +25

You need write only one program for a whole family of parts, entering the characteristic dimensions as Q parameters.

To program a particular part, you then assign the appropriate values to the individual Q parameters.

Example: Cylinder with Q parameters

Cylinder radius:	R = Q1
Cylinder height:	H = Q2
Cylinder Z1:	Q1 = +30 Q2 = +10
Cylinder Z2:	Q1 = +10 Q2 = +50



9.3 Describing contours with mathematical functions

Application

The Q parameters listed below enable you to program basic mathematical functions in a NC program:

- Select a Q parameter function: Press the **Q** key (in the numerical keypad on the right). The Q parameter functions are displayed in a soft key row
- Select the basic mathematical functions by pressing the **BASIC ARITHM...** soft key
- > The control then displays the following soft keys:

Overview

Soft key	Function
FN0 X = Y	**FN 0**: ASSIGN e. g., **FN 0: Q5 = +60** Directly assign value Reset Q parameter value
FN1 X + Y	**FN 1**: ADDITION e. g., **FN 1: Q1 = -Q2 + -5** Calculate and assign the sum of two values
FN2 X - Y	**FN 2**: SUBTRACTION e. g. **FN 2: Q1 = +10 - +5** Form and assign difference between two values
FN3 X * Y	**FN 3**: MULTIPLICATION e. g. **FN 3: Q2 = +3 * +3** Form and assign the product of two values
FN4 X / Y	**FN 4**: DIVISION e.g., **FN 4: Q4 = +8 DIV +Q2** Calculate and assign the quotient of two values **Not permitted:** Division by 0
FN5 SQRT	**FN 5**: SQUARE ROOT e.g., **FN 5: Q20 = SQRT 4** Calculate and assign the square root of a value **Not permitted:** Square root of a negative value

You can enter the following to the right of the **=** sign:

- Two numbers
- Two Q parameters
- A number and a Q parameter

The Q parameters and numerical values in the equations can be entered with positive or negative signs.

Programming fundamental operations

ASSIGN

Example

```
16 FN 0: Q5 = +10
17 FN 3: Q12 = +Q5 * +7
```

Q

- Select the Q parameter function: Press the **Q** key

BASIC ARITHM.

- Select basic mathematical functions by pressing the **BASIC ARITHM.** soft key

FN0 X = Y

- To select the ASSIGN Q parameter function: Press the **FN 0 X = Y** soft key

PARAMETER NUMBER FOR RESULT?

ENT

- Enter **5** (the number of the Q parameter) and confirm with the **ENT** key

FIRST VALUE / PARAMETER?

ENT

- Enter **10**: Assign the numerical value 10 to Q5 and confirm with the **ENT** key

MULTIPLICATION

Q

- Select the Q parameter function: Press the **Q** key

BASIC ARITHM.

- Select basic mathematical functions by pressing the **BASIC ARITHM.** soft key

FN3 X * Y

- To select the MULTIPLICATION Q parameter function, press the **FN 3 X * Y** soft key

PARAMETER NUMBER FOR RESULT?

ENT

- Enter **12** (the number of the Q parameter) and confirm with the **ENT** key

FIRST VALUE / PARAMETER?

ENT

- Enter **Q5** as the first value and confirm with the **ENT** key.

SECOND VALUE / PARAMETER?

ENT

- Enter **7** as the second value and confirm with the **ENT** key.

Resetting Q parameters

Example

```
16 FN 0: Q5 SET UNDEFINED
17 FN 0:  Q1 = Q5
```

Q
- Select the Q parameter function: Press the **Q** key

BASIC ARITHM.
- Select basic mathematical functions by pressing the **BASIC ARITHM.** soft key

FN0 X = Y
- To select the ASSIGN Q parameter function: Press the **FN 0 X = Y** soft key

PARAMETER NUMBER FOR RESULT?

ENT
- Enter **5** (the number of the Q parameter) and confirm with the **ENT** key

1. VALUE OR PARAMETER?

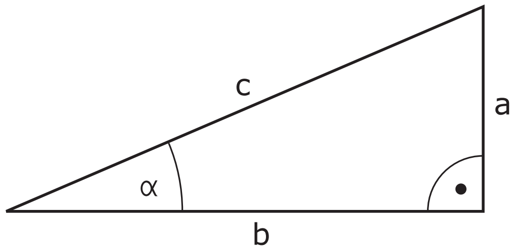
SET UNDEFINED
- Press **SET UNDEFINED**

> The **FN 0** function also supports transfer of the value **Undefined**. If you wish to transfer the undefined Q parameter without **FN 0**, the control shows the error message **Invalid value**.

9.4 Trigonometric functions

Definitions

Sine: sin α = a / c
Cosine: cos α = b / c
Tangent: tanα = a / b = sin α / cos α



where

- c is the side opposite the right angle
- a is the side opposite the angle α
- b is the third side.

The control can find the angle from the tangent:

α = arctan (a / b) = arctan (sin α / cos α)

Example:

a = 25 mm

b = 50 mm

α = arctan (a / b) = arctan 0.5 = 26.57°

Furthermore:

$a^2 + b^2 = c^2$ (where $a^2 = a \times a$)

$c = \sqrt{(a^2 + b^2)}$

Programming trigonometric functions

The angle functions appear when the **TRIGONOMETRY** soft key is pressed. The control displays the soft keys listed in the table below.

Soft key	Function
FN6 SIN(X)	**FN 6**: SINUS e. g., **FN 6: Q20 = SIN-Q5** Calculate and assign the sine of an angle in degrees (°)
FN7 COS(X)	**FN 7**: COSINE e. g., **FN 7: Q21 = COS-Q5** Calculate and assign the cosine of an angle in degrees (°)
FN8 X LEN Y	**FN 8**: ROOT SUM OF SQUARES e. g., **FN 8: Q10 = +5 LEN +4** Calculate and assign lengths from two values
FN13 X ANG Y	**FN 13**: ANGLE e. g., **FN 13: Q20 = +25 ANG-Q1** Calculate and assign an angle with the arc tangent from the opposite and adjacent sides or with the sine and cosine of the angle (0 < angle < 360°)

9.5 Circle calculations

Application

The control can use the functions for calculating circles to calculate the circle center and the circle radius from three or four given points on the circle. The calculation is more accurate if four points are used.

Application: These functions can be used, for example, if you wish to determine the location and size of a hole or a pitch circle using the programmable probing function.

Soft key	Function
FN23 3 POINTS OF CIRCLE	FN 23: Determining the CIRCLE DATA from three points e. g., **FN 23: Q20 = CDATA Q30**

The coordinate pairs of three points on a circle must be saved in Q30 and the following five parameters—in this case, up to Q35.

The control then saves the circle center in the reference axis (X if spindle axis is Z) in parameter Q20, the circle center in the minor axis (Y if spindle axis is Z) in parameter Q21, and the circle radius in parameter Q22.

Soft key	Function
FN24 4 POINTS OF CIRCLE	FN 24: Determining the CIRCLE DATA from four points e. g., **FN 24: Q20 = CDATA Q30**

The coordinate pairs of four points on a circle must be saved in Q30 and the following seven parameters—in this case, up to Q37.

The control then saves the circle center in the reference axis (X if spindle axis is Z) in parameter Q20, the circle center in the minor axis (Y if spindle axis is Z) in parameter Q21, and the circle radius in parameter Q22.

> Note that **FN 23** and **FN 24** automatically overwrite the resulting parameter and the two following parameters.

9.6 If-then decisions with Q parameters

Application

In if-then decisions, the control compares a Q parameter with another Q parameter or a numerical value. If the condition is fulfilled, then the control continues the NC program at the label that is programmed after the condition.

> Before creating your NC program, compare the if-then decisions with the subprogram and program section repeat programming techniques.
>
> You can thereby avoid possible misunderstandings and programming errors.
>
> **Further information:** "Labeling subprograms and program section repeats", Page 244

If it is not fulfilled, the control continues with the next NC block.

If you want to call an NC program, then program a program call with **PGM CALL** after the label.

Unconditional jumps

An unconditional jump is programmed by entering a conditional jump whose condition is always true. Example:

FN 9: IF+10 EQU+10 GOTO LBL1

Conditioning jumps with counters

The jump function allows you to repeat a machining operation any number of times. A Q parameter serves as a counter that increments by 1 at every program section repeat.

The jump function allows you to compare the counter with the number of desired machining operations.

> These jumps differ from the subprogram and program section repeat programming techniques.
>
> On the one hand, for example, jumps require no completed program section ending with LBL 0. On the other hand, jumps do not take these return jump labels into consideration!

Example

0 BEGIN PGM COUNTER MM	
1 ;	
2 Q1 = 0	Loaded value: Initialize counter
3 Q2 = 3	Loaded value: Number of jumps
4 ;	
5 LBL 99	Label
6 Q1 = Q1 + 1	Initialize counter: New Q1 value = Old Q1 value + 1
7 FN 12: IF +Q1 LT +Q2 GOTO LBL 99	Run program jumps 1 and 2
8 FN 9: IF +Q1 EQU +Q2 GOTO LBL 99	Run program jump 3
9 ;	
10 END PGM COUNTER MM	

Abbreviations used

IF	:	If
EQU	:	Equal to
NE	:	Not equal to
GT	:	Greater than
LT	:	Less than
GOTO	:	Go to
UNDEFINED	:	Undefined
DEFINED	:	Defined

Programming if-then decisions

Possibilities for jump inputs

The following inputs are possible for the condition **IF**:

- Numbers
- Texts
- Q, QL, QR
- **QS** (string parameter)

You have three possibilities for entering the jump address **GOTO**:

- **LBL NAME**
- **LBL NUMBER**
- **QS**

The if-then decisions appear when the **JUMP** soft key is pressed. The control displays the following soft keys:

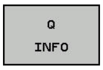
Soft key	Function
FN9 IF X EQ Y GOTO EQU	**FN 9**: IF EQUAL, JUMP e. g. **FN 9: IF +Q1 EQU +Q3 GOTO LBL "UPCAN25"** If both values or parameters are equal, jump to specified label
FN9 IF X EQ Y GOTO IS UNDEFINED	**FN 9**: IF UNDEFINED, JUMP e. g., **FN 9: IF +Q1 IS UNDEFINED GOTO LBL “UPCAN25“** If the specified parameter is undefined, then a jump is made to the specified label
FN9 IF X EQ Y GOTO IS DEFINED	**FN 9**: IF DEFINED, JUMP e. g., **FN 9: IF +Q1 IS DEFINED GOTO LBL “UPCAN25“** If the specified parameter is defined, then a jump is made to the specified label
FN10 IF X NE Y GOTO	**FN 10**: IF UNEQUAL, JUMP e. g.**FN 10: IF +10 NE -Q5 GOTO LBL 10** If both values or parameters are unequal, jump to specified label
FN11 IF X GT Y GOTO	**FN 11**: IF GREATER, JUMP g. g.**FN 11: IF+Q1 GT+10 GOTO LBL QS5** If the first value or parameter is greater than the second value or parameter, jump to specified label
FN12 IF X LT Y GOTO	**FN 12**: IF LESS, JUMP e. g. **FN 12: IF+Q5 LT+0 GOTO LBL "ANYNAME"** If the first value or parameter is smaller than the second value or parameter, jump to specified label

9.7 Checking and changing Q parameters

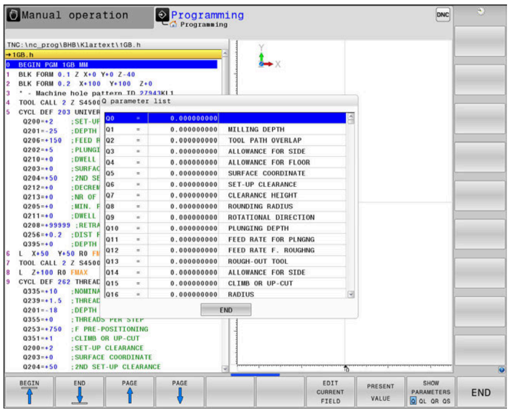
Procedure

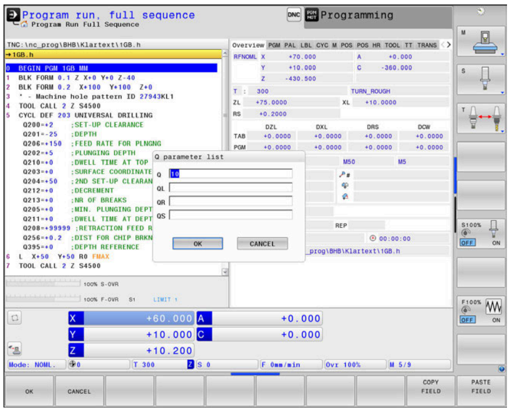
You can check Q parameters in all operating modes, and also edit them.

- If you are in a program run, interrupt it if required (e.g. by pressing the **NC STOP** key and the **INTERNAL STOP** soft key) or stop the test run



- To call the Q parameter functions, press the **Q INFO** soft key or the **Q** key
- The control lists all of the parameters and their corresponding current values.
- Use the arrow keys or the **GOTO** key to select the desired parameter.
- To change the value, press the **EDIT CURRENT FIELD** soft key, enter the new value, and confirm with the **ENT** key
- To leave the value unchanged, press the **PRESENT VALUE** soft key or close the dialog with the **END** key





> All of the parameters with displayed comments are used by the control within cycles or as transfer parameters.
>
> If you want to check or edit local, global or string parameters, press the **SHOW PARAMETERS Q QL QR QS** soft key. The control then displays the specific parameter type. The functions previously described also apply.

You can have Q parameters also displayed in the additional status display in all operating modes (except **Programming** mode).

- If you are in a program run, interrupt it if required (e.g. by pressing the **NC STOP** key and the **INTERNAL STOP** soft key) or stop the test run
- Display the soft key row for screen layout

PROGRAM + STATUS

- Select the layout option for the additional status display
- > In the right half of the screen, the control shows the **Overview** status form.

STATUS OF Q PARAM.

- Press the **STATUS OF Q PARAM.** soft key

Q PARAMETER LIST

- Press the **Q PARAMETER LIST** soft key
- > The control opens a pop-up window.
- For each parameter type (Q, QL, QR, QS), define the parameter numbers you wish to check. Separate single Q parameters with a comma, and connect sequential Q parameters with a hyphen, e.g. 1,3,200-208. The input range per parameter type is 132 characters

> The display in the **QPARA** tab always contains eight decimal places. The result of Q1 = COS 89.999 is shown by the control as 0.00001745, for example. Very large or very small values are displayed by the control in exponential notation. The result of Q1 = COS 89.999 * 0.001 is shown by the control as +1.74532925e-08, whereby e-08 corresponds to the factor of 10^{-8}.

9.8 Additional functions

Overview

The additional functions appear when the **DIVERSE FUNCTION** soft key is pressed. The control displays the following soft keys:

Soft key	Function	Page
FN14 ERROR=	**FN 14: ERROR** Display error messages	281
FN16 F-PRINT	**FN 16: F-PRINT** Formatted output of texts or Q parameter values	285
FN18 SYS-DATUM READ	**FN 18: SYSREAD** Read system data	293
FN19 PLC=	**FN 19: PLC** Transfer values to the PLC	293
FN20 WAIT FOR	**FN 20: WAIT FOR** NC and PLC synchronization	294
FN26 OPEN TABLE	**FN 26: TABOPEN** Open a freely definable table	390
FN27 WRITE TO TABLE	**FN 27: TABWRITE** Write to a freely definable table	390
FN28 READ FROM TABLE	**FN 28: TABREAD** Read from a freely definable table	391
FN29 PLC LIST=	**FN 29: PLC** Transfer up to eight values to the PLC	295
FN37 EXPORT	**FN 37: EXPORT** Export local Q parameters or QS parameters into a calling NC program	296
FN38 SEND	**FN 38: SEND** Send information from the NC program	296

FN 14: ERROR – Displaying error messages

With the **FN 14: ERROR** error function, you can output error messages under program control. The messages are predefined by the machine tool builder or by HEIDENHAIN. If the control encounters an NC block with **FN 14: ERROR** during program run, it will interrupt the run and display an error message. You must then restart the NC program.

Error numbers area	Standard dialog
0 ... 999	Machine-dependent dialog
1000 ... 1199	Internal error messages

Example

The control is intended to display a message if the spindle is not switched on.

```
180 FN 14: ERROR = 1000
```

Error message predefined by HEIDENHAIN

Error number	Text
1000	Spindle?
1001	Tool axis is missing
1002	Tool radius too small
1003	Tool radius too large
1004	Range exceeded
1005	Start position incorrect
1006	ROTATION not permitted
1007	SCALING FACTOR not permitted
1008	MIRROR IMAGE not permitted
1009	Datum shift not permitted
1010	Feed rate is missing
1011	Input value incorrect
1012	Incorrect sign
1013	Entered angle not permitted
1014	Touch point inaccessible
1015	Too many points
1016	Contradictory input
1017	CYCL incomplete
1018	Plane wrongly defined
1019	Wrong axis programmed
1020	Wrong rpm
1021	Radius comp. undefined

Error number	Text
1022	Rounding-off undefined
1023	Rounding radius too large
1024	Program start undefined
1025	Excessive nesting
1026	Angle reference missing
1027	No fixed cycle defined
1028	Slot width too small
1029	Pocket too small
1030	Q202 not defined
1031	Q205 not defined
1032	Q218 must be greater than Q219
1033	CYCL 210 not permitted
1034	CYCL 211 not permitted
1035	Q220 too large
1036	Q222 must be greater than Q223
1037	Q244 must be greater than 0
1038	Q245 must not equal Q246
1039	Angle range must be under 360°
1040	Q223 must be greater than Q222
1041	Q214: 0 not permitted
1042	Traverse direction not defined
1043	No datum table active
1044	Position error: center in axis 1
1045	Position error: center in axis 2
1046	Hole diameter too small
1047	Hole diameter too large
1048	Stud diameter too small
1049	Stud diameter too large
1050	Pocket too small: rework axis 1
1051	Pocket too small: rework axis 2
1052	Pocket too large: scrap axis 1
1053	Pocket too large: scrap axis 2
1054	Stud too small: scrap axis 1
1055	Stud too small: scrap axis 2
1056	Stud too large: rework axis 1
1057	Stud too large: rework axis 2

Error number	Text
1058	TCHPROBE 425: length exceeds max
1059	TCHPROBE 425: length below min
1060	TCHPROBE 426: length exceeds max
1061	TCHPROBE 426: length below min
1062	TCHPROBE 430: diameter too large
1063	TCHPROBE 430: diameter too small
1064	No measuring axis defined
1065	Tool breakage tolerance exceeded
1066	Enter Q247 unequal to 0
1067	Enter Q247 greater than 5
1068	Datum table?
1069	Enter Q351 unequal to 0
1070	Thread depth too large
1071	Missing calibration data
1072	Tolerance exceeded
1073	Block scan active
1074	ORIENTATION not permitted
1075	3-D ROT not permitted
1076	Activate 3-D ROT
1077	Enter depth as negative
1078	Q303 in meas. cycle undefined!
1079	Tool axis not allowed
1080	Calculated values incorrect
1081	Contradictory meas. points
1082	Incorrect clearance height
1083	Contradictory plunge type
1084	This fixed cycle not allowed
1085	Line is write-protected
1086	Oversize greater than depth
1087	No point angle defined
1088	Contradictory data
1089	Slot position 0 not allowed
1090	Enter an infeed not equal to 0
1091	Switchover of Q399 not allowed
1092	Tool not defined
1093	Tool number not permitted

Error number	Text
1094	Tool name not permitted
1095	Software option not active
1096	Kinematics cannot be restored
1097	Function not permitted
1098	Contradictory workpc. blank dim.
1099	Measuring position not allowed
1100	Kinematic access not possible
1101	Meas. pos. not in traverse range
1102	Preset compensation not possible
1103	Tool radius too large
1104	Plunging type is not possible
1105	Plunge angle incorrectly defined
1106	Angular length is undefined
1107	Slot width is too large
1108	Scaling factors not equal
1109	Tool data inconsistent

FN 16: F-PRINT – Formatted output of text and Q parameter values

Basics

With the function **FN 16: F-PRINT**, you can save Q parameter values and output formatted texts (e.g. in order to save measurement reports).

You can output the values as follows:

- Save them to a file on the control
- Display them on the screen in a pop-up window
- Save them to an external file
- Print them using a connected printer

Procedure

Proceed as follows in order to output Q-parameter values and texts:

- Create a text file that defines the output format and contents
- In the NC program, use the function **FN 16: F-PRINT** in order to output the log

If you output the values to a file, the maximum size if the output file will be 20 KB.

Changing the output path of the log file

If you wish to save the measurement results in another directory, then you must modify the output path of the log file.

To change the output path, proceed as follows:

MOD

- Press the **MOD** key
- Enter the code number 123
- Select the parameter **Paths for the end user** (**CfgUserPath**)
- Select the parameter **FN 16 output path for execution** (**fn16DefaultPath**)
- > The control opens a pop-up window.
- Select the output path for the machine operating modes
- Select the parameter **FN 16 output path for the Programming and Test Run op. modes** (**fn16DefaultPathSim**)
- > The control opens a pop-up window.
- Select the output path for the **Programming** and **Test Run** operating modes

Creating a text file

To output the formatted texts and Q parameter values, use the control's text editor to create a text file. Define the format and Q parameters to be output in this file.

Proceed as follows:

PGM MGT

- Press the **PGM MGT** key

NEW FILE

- Press the **NEW FILE** soft key
- Create a file with the extension **.A**

Available functions

Use the following formatting functions for creating a text file:

Special characters	Function
"..........."	Define output format for texts and variables between the quotation marks
%F	Format for Q parameters, QL, and QR: ■ Define %: format ■ F: Floating (decimal number), format for Q, QL, QR
9.3	Format for Q parameters, QL, and QR: ■ Total of 9 characters, including decimal separator ■ Of these, 3 are decimal places
%S	Format for text variable QS
%RS	Format for text variable QS Assumes the subsequent without any changes or formatting
%D or **%I**	Format for integer
,	Separation character between output format and parameter
;	End of block character
*****	Beginning of a comment line Comments are not shown in the log
%"	Output quotation marks
%%	Output percent sign
****	Output backslash
\n	Output line break
+	Q parameter value, right-aligned
-	Q parameter value, left-aligned

Example

Input	Meaning
"X1 = %+9.3F", Q31;	Format for Q parameter: ■ **"X1 =**: The text **X1 =** is output ■ **%**: Specify the format ■ **+**: Number right-aligned ■ **9.3**: Total of 9 characters; 3 of them are decimal places ■ **F**: Floating (decimal number) ■ **, Q31**: Output the value from Q31 ■ **;**: End of block

The following functions allow you to include the following additional information in the protocol log file:

Keyword	Function
CALL_PATH	Gives the path for the NC program where you will find the FN 16 function. Example: "Measuring program: %S",CALL_PATH;
M_CLOSE	Closes the file to which you are writing with FN 16. Example: M_CLOSE;
M_APPEND	Upon renewed output, appends the log to the existing log. Example: M_APPEND;
M_APPEND_MAX	Upon renewed output, appends the log to the existing log until the maximum specified file size in kilobytes is exceeded. Example: M_APPEND_MAX20;
M_TRUNCATE	Overwrites the log upon renewed output. Example: M_TRUNCATE;
L_ENGLISH	Outputs the text only if English is set as dialog language
L_GERMAN	Outputs the text only if German is set as dialog language
L_CZECH	Outputs text only for Czech conversational language
L_FRENCH	Outputs text only for French conversational language
L_ITALIAN	Outputs text only for Italian conversational language
L_SPANISH	Outputs text only for Spanish conversational language
L_PORTUGUE	Outputs text only for Portuguese conversational language
L_SWEDISH	Outputs text only for Swedish conversational language
L_DANISH	Outputs text only for Danish conversational language
L_FINNISH	Outputs text only for Finnish conversational language
L_DUTCH	Outputs text only for Dutch conversational language
L_POLISH	Outputs text only for Polish conversational language
L_HUNGARIA	Outputs text only for Hungarian conversational language
L_CHINESE	Outputs text only for Chinese conversational language
L_CHINESE_TRAD	Outputs text only for Chinese (traditional) conversational language

Keyword	Function
L_SLOVENIAN	Outputs text only for Slovenian conversational language
L_NORWEGIAN	Outputs text only for Norwegian conversational language
L_ROMANIAN	Outputs text only for Romanian conversational language
L_SLOVAK	Outputs text only for Slovakian conversational language
L_TURKISH	Outputs text only for Turkish conversational language
L_ALL	Display text independently of the conversational language
HOUR	Number of hours from the real-time clock
MIN	Number of minutes from the real-time clock
SEC	Number of seconds from the real-time clock
DAY	Day from the real-time clock
MONTH	Month as a number from the real-time clock
STR_MONTH	Month as a string abbreviation from the real-time clock
YEAR2	Two-digit year from the real-time clock
YEAR4	Four-digit year from the real-time clock

Example

Example of a text file to define the output format:

```
"MEASURING LOG OF IMPELLER CENTER OF GRAVITY";
"DATUM: %02d.%02d.%04d",DAY,MONTH,YEAR4;
"TIME: %02d:%02d:%02d",HOUR,MIN,SEC;
"NO. OF MEASURED VALUES: = 1";
"X1 = %9.3F", Q31;
"Y1 = %9.3F", Q32;
"Z1 = %9.3F", Q33;
L_GERMAN;
"Werkzeuglänge beachten";
L_ENGLISH;
"Remember the tool length";
```

Activating FN 16 output in an NC program

Within the **FN 16** you specify the output file that contains the texts to be output.

The control generates the output file:

- at the end of the program (**END PGM**),
- if a program is canceled (**NC STOP** key)
- as a result of the command **M_CLOSE**

Enter the path of the source and the path of the output file in the FN 16 function .

Proceed as follows:

Q

- Press the **Q** key.

DIVERSE FUNCTION

- Press the **DIVERSE FUNCTION** soft key

FN16 F-PRINT

- Press the **FN16 F-PRINT** soft key

SELECT FILE

- Press the **SELECT FILE** soft key
- Select the source, i.e. the text file in which the output file is defined

ENT

- Confirm with the **ENT** key
- Enter the output path.

Path entries in the FN 16 function

If you enter only the file name as the path for the log file, the control saves the log file in the directory in which the NC program with the **FN 16** function is located.

Program relative paths as an alternative to complete paths:

- Starting from the folder of the calling file one folder level down **FN 16: F-PRINT MASKE\MASKE1.A/ PROT\PROT1.TXT**
- Starting from the folder of calling file one folder level up and in another folder **FN 16: F-PRINT ..\MASKE\MASKE1.A/ .. \PROT1.TXT**

Operating and programming notes:

- If you output the same file more than once in the NC program, the control appends the current output to the end of the contents already output within the target file.
- In the **FN 16** block, program the format file and the log file, each with the extension for the file type.
- The file name extension of the log file determines the file type of the output (e.g., TXT, A, XLS, HTML).
- If you use **FN 16**, then no UTF-8 encoding is permitted for the file.
- Use **FN 18** to receive much information that is relevant and interesting in log files, such as the number of the touch-probe cycle last used. **Further information:** "FN 18: SYSREAD – Reading system data", Page 293

Enter the source or the target with parameters

You can enter the source file and the output file as Q parameters or as QS parameters. For this purpose you previously define the desired parameter in the NC program.

Further information: "Assign string parameters", Page 325

Enter Q parameters in the **FN 16** function with the following syntax so that the control can detect the Q parameters:

Input	Function
:'QS1'	Set QS parameters with preceding colon and between single quotation marks
:'QL3'.txt	Specify additional file name extension for the target file if required

> If you want to output a path with a QS parameter to a log file, then use the function **%RS**. This ensures that the control does not interpret the special characters as formatting characters.

Example

```
96 FN 16: F-PRINT TNC:\MASKE\MASKE1.A/ TNC:\PROT1.TXT
```

The control creates the file PROT1.TXT:

```
MEASURING LOG OF IMPELLER CENTER OF GRAVITY
DATE: July 15, 2015
TIME: 8:56:34 AM
NO. OF MEASURED VALUES : = 1
X1 = 149.360
Y1 = 25.509
Z1 = 37.000
Remember the tool length
```

Displaying messages on the control screen

You can also use the function **FN 16: F-PRINT** to display any messages from the NC program in a pop-up window on the control screen. This makes it easy to display explanatory texts, including long texts, at any point in the NC program in a way that the user has to react to them. You can also display Q-parameter contents if the protocol description file contains such instructions.

For the message to appear on the control screen, you need only enter **SCREEN:** as the output path.

Example

```
96 FN 16: F-PRINT TNC:\MASKE\MASKE1.A/SCREEN:
```

If the message has more lines than fit in the pop-up window, you can use the arrow keys to page in the window.

> If you output the same file more than once in the NC program, the control appends the current output to the end of the contents already output within the target file.
>
> If you want to overwrite the previous pop-up window, program the function **M_CLOSE** or **M_TRUNCATE**.

Close the pop-up window

You can close the pop-up window in the following ways:

- Press the **CE** key
- Controlled by the program with the output path **sclr:**

Example

```
96 FN 16: F-PRINT TNC:\MASKE\MASKE1.A/SCLR:
```

Exporting messages

With the **FN 16** function you can also store log files externally.

To do so you must enter the target path in the **FN 16** function.

Example

```
96 FN 16: F-PRINT TNC:\MSK\MSK1.A / PC325:\LOG\PRO1.TXT
```

> If you output the same file more than once in the NC program, the control appends the current output to the end of the contents already output within the target file.

Printing messages

You can also use the function **FN 16: F-PRINT** to print any messages on a connected printer.

Further information: User's Manual for Setup, Testing and Running NC Programs

In order for the messages to be sent to the printer, you must enter **Printer:** as the name of the log file and then enter the corresponding file name.

The control saves the file in the **PRINTER:** path until the file is printed.

Example

```
96 FN 16: F-PRINT TNC:\MASKE\MASKE1.A/PRINTER:\DRUCK1
```

FN 18: SYSREAD – Reading system data

With the **FN 18: SYSREAD** function, you can read system data and save them to Q parameters. The selection of the system datum occurs via a group number (ID no.), a system data number, and, if necessary, an index.

> The read values of the function **FN 18: SYSREAD** are always output by the control in **metric** units regardless of the NC program’s unit of measure.

Further information: "System data", Page 558

Example: Assign the value of the active scaling factor for the Z axis to Q25.

```
55 FN 18: SYSREAD Q25 = ID210 NR4 IDX3
```

FN 19: PLC – Transfering values to the PLC

NOTICE

Danger of collision!

Changes to the PLC can result in undesired behavior and serious errors (e.g., inoperability of the control). For this reason, access to the PLC is protected by password. The FN function provides HEIDENHAIN as well as your machine tool builder and suppliers the ability to communicate with the PLC from an NC program. It is not recommended that the machine operator or NC programmer use this. There is risk of collision during the execution of the function and during the subsequent processing!

- Only use the function in consultation with HEIDENHAIN, the machine tool builder, or the supplier.
- Comply with the documentation from HEIDENHAIN, the machine tool builder, and suppliers.

The **FN 19: PLC** function transfers up to two numerical values or Q parameters to the PLC.

FN 20: WAIT FOR – NC and PLC synchronization

NOTICE

Danger of collision!

Changes to the PLC can result in undesired behavior and serious errors (e.g., inoperability of the control). For this reason, access to the PLC is protected by password. The FN function provides HEIDENHAIN as well as your machine tool builder and suppliers the ability to communicate with the PLC from an NC program. It is not recommended that the machine operator or NC programmer use this. There is risk of collision during the execution of the function and during the subsequent processing!

- Only use the function in consultation with HEIDENHAIN, the machine tool builder, or the supplier.
- Comply with the documentation from HEIDENHAIN, the machine tool builder, and suppliers.

With the **FN 20: WAIT FOR** function you can synchronize the NC and PLC during a program run. The NC stops machining until the condition that you have programmed in the **FN 20: WAIT FOR** block is fulfilled.

SYNC is used whenever you read, for example, system data via **FN 18: SYSREAD** that require synchronization with real time. The control stops the look-ahead calculation and executes the following NC block only when the NC program has actually reached that NC block.

Example: Pause internal look-ahead calculation, read current position in the X axis

```
32 FN 20: WAIT FOR SYNC
33 FN 18: SYSREAD Q1 = ID270 NR1 IDX1
```

FN 29: PLC – Transferring values to the PLC

NOTICE

Danger of collision!

Changes to the PLC can result in undesired behavior and serious errors (e.g., inoperability of the control). For this reason, access to the PLC is protected by password. The FN function provides HEIDENHAIN as well as your machine tool builder and suppliers the ability to communicate with the PLC from an NC program. It is not recommended that the machine operator or NC programmer use this. There is risk of collision during the execution of the function and during the subsequent processing!

- Only use the function in consultation with HEIDENHAIN, the machine tool builder, or the supplier.
- Comply with the documentation from HEIDENHAIN, the machine tool builder, and suppliers.

The **FN 29: PLC** function transfers up to eight numerical values or Q parameters to the PLC.

FN 37: EXPORT

NOTICE

Danger of collision!

Changes to the PLC can result in undesired behavior and serious errors (e.g., inoperability of the control). For this reason, access to the PLC is protected by password. The FN function provides HEIDENHAIN as well as your machine tool builder and suppliers the ability to communicate with the PLC from an NC program. It is not recommended that the machine operator or NC programmer use this. There is risk of collision during the execution of the function and during the subsequent processing!

- Only use the function in consultation with HEIDENHAIN, the machine tool builder, or the supplier.
- Comply with the documentation from HEIDENHAIN, the machine tool builder, and suppliers.

You need the **FN 37: EXPORT** function if you want to create your own cycles and integrate them in the control.

FN 38: SEND – Send information from the NC program

The function **FN 38: SEND** enables you to retrieve texts and Q parameter values from the NC program and write them to the log or send them to an external application, e.g. the StateMonitor.

The syntax consists of two parts:

- **Format of transmitted text**: Output text with optional placeholders for variable values, e.g. **%f**

  > Input may be in the form of QS parameters.
  >
  > Placeholders are case-sensitive, so make sure to enter them correctly.

- **Datum for placeholder in text**: List of max. 7 Q, QL, or QR-Variables, e.g. **Q1**

Data transmission is through a standard TCP/IP computer network.

> For more detailed information, consult the RemoTools SDK manual.

Example

Document the values from Q1 and Q23 in the log.

```
FN 38: SEND /"Q-Parameter Q1: %f Q23: %f" / +Q1 / +Q23
```

Example

Define the output format for the variable values.

```
FN 38: SEND /"Q-Parameter Q1: %05.1f" / +Q1
```

> The control outputs the variable value as a 5-digit number, one of them being a decimal place. If required, leading zeros will be added to pad the field.

```
FN 38: SEND /"Q-Parameter Q1: % 1.3f" / +Q1
```

> The control outputs the variable value as a 7-digit number, three of them being decimal places. If required, blanks will be added to pad the field.

To obtain **%** in the output text, enter **%%** at the desired position.

Example

Send information to the StateMonitor.

With function **FN 38**, you can enter job data, among others. For this purpose, a job must have been created in the StateMonitor and it must have been assigned to the machine tool to be used.

> Job management is possible with StateMonitor version 1.2 or higher using the so-called JobTerminals (Option 4).

Requirements:

- Job number 1234
- Working step 1

FN 38: SEND /"JOB:1234_STEP:1_CREATE"	Create job
FN 38: SEND /"JOB:1234_STEP:1_CREATE_ITEMNAME:HOLDER_I	Alternative: Create job with part name, part number, and required quantity
FN 38: SEND /"JOB:1234_STEP:1_START"	Start job
FN 38: SEND /"JOB:1234_STEP:1_PREPARATION"	Start preparation
FN 38: SEND /"JOB:1234_STEP:1_PRODUCTION"	Production
FN 38: SEND /"JOB:1234_STEP:1_STOP"	Stop job
FN 38: SEND /"JOB:1234_STEP:1_ FINISH"	Finish job

In addition, the workpiece quantities for the job can be returned.

With the **OK**, **S** (for Scrap), and **R** (for Rework) placeholders, you can specify whether the reported workpieces have been machined correctly or not.

The **A** and **I** placeholders, you can define how the StateMonitor will handle these reports. If absolute values are passed, the StateMonitor will overwrite the previously valid values. If incremental values are used, the StateMonitor will increment the qualities.

FN 38: SEND /"JOB:1234_STEP:1_OK_A:23"	Actual amount (OK) absolute value
FN 38: SEND /"JOB:1234_STEP:1_OK_I:1"	Actual amount (OK) incremental value
FN 38: SEND /"JOB:1234_STEP:1_S_A:12"	Scrap (S) absolute value
FN 38: SEND /"JOB:1234_STEP:1_S_I:1"	Scrap (S) incremental value
FN 38: SEND /"JOB:1234_STEP:1_R_A:15"	Rework (R) absolute value
FN 38: SEND /"JOB:1234_STEP:1_R_I:1"	Rework (R) incremental value

9.9 Accessing tables with SQL commands

Introduction

If you would like to access numerical or alphanumerical content in a table or manipulate the table (e.g., rename columns or rows), then use the SQL commands available to you.

The syntax of the SQL commands available on the control is heavily influenced by the SQL programming language—but does not conform to it completely. In addition, the control does not support the entire scope of the SQL language.

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

> SQL functions can only be tested in the **Program run, single block**, **Program run, full sequence** and **Positioning with Manual Data Input** operating mode.

> Read- and write-access to individual values of a table can likewise be carried out using the function **FN 26: TABOPEN**, **FN 27: TABWRITE**, and **FN 28: TABREAD**.
> **Further information:** "Freely definable tables", Page 387
>
> HEIDENHAIN recommends using SQL functions instead of **FN 26**, **FN 27**, or **FN 28** with HDR hard disks in order to achieve maximum speeds with table applications and also to reduce the amount of computing power necessary.

The following terms will be used (along with others) in the following:

- “SQL command” refers to the available soft keys
- “SQL instructions” describe miscellaneous functions that are entered manually as part of the syntax
- **HANDLE** in the syntax identifies a certain transaction (followed by the parameter for identification)
- **Result-set** contains the result of the query (known as the result set)

SQL transaction

In the NC software, access to tables is gained via an SQL server. This server is controlled with the available SQL commands. The SQL commands can be defined directly in an NC program.

The saver is based on a transaction model. A **transaction** consists of multiples steps that are executed together, thereby ensuring an orderly and defined processing of the table entries.

Example of transaction:

- Assign Q parameters to table columns for read or write access using **SQL BIND**
- Select data using **SQL EXECUTE** with the instruction **SELECT**
- Read, change, or add data using **SQL FETCH**, **SQL UPDATE**, or **SQL INSERT**
- Confirm or discard interaction using **SQL COMMIT** or **SQL ROLLBACK**
- Approve bindings between table columns and Q parameters using **SQL BIND**

> You must conclude all transactions that have been started—even exclusively reading accesses. Concluding the transaction is the only way to ensure that changes and additions are transferred, that locks are removed, and that used resources are released.

Result set and handle

The **result set** contains a subset of a table file. It results from a **SELECT** query performed on the table.

The **result set** is created when a query is executed in the SQL server, thereby occupying resources there.

This query has the same effect as applying a filter to the table, so that only part of the data records become visible. To perform this query, the table file must be read at this point.

The SQL server assigns a **handle** to the **result set,** which enables you to identify the result set for reading/editing data and completing the transaction. The **handle** is the result of the query, which is visible in the NC program. The value 0 indicates an **invalid handle**, i.e. it was not possible to create a **result set** for that query. If no rows are found that satisfy the specified condition, an empty **result set** is created and assigned a valid **handle.**

Programming SQL commands

> This function is not enabled until the code number **555343** is entered.

You can program SQL commands in the **Programming** or **Positioning with mdi** operating modes:

SPEC FCT

- Press the **SPEC FCT** key

PROGRAM FUNCTIONS

- Press the **PROGRAM FUNCTIONS** soft key

▷

- Shift the soft-key row

SQL

- Press the **SQL** soft key
- Select the SQL command via soft key

NOTICE

Danger of collision!

Read and write accesses performed with the help of SQL commands always occur in metric units, regardless of the unit of measure selected for the table or the NC program.
If, for example, you save a length from a table to a Q parameter, then the value is thereafter always in metric units. If this value is then used for the purpose of positioning in an inch program (**L X +Q1800**), then an incorrect position will result.

- In inch programs, convert the read value prior to use

Overview of functions

Overview of soft keys

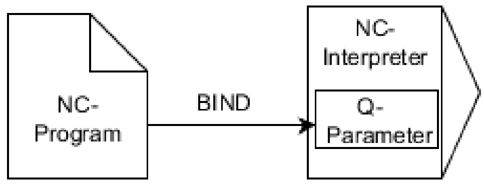
The control offers the following ways of working with SQL commands:

Soft key	Command	Page
SQL BIND	**SQL BIND** creates or disconnects a binding between table columns and Q or QS parameters	303
SQL EXECUTE	**SQL EXECUTE** opens a transaction for selected table columns and table rows or enables the use of other SQL instructions (miscellaneous functions).	304
SQL FETCH	**SQL FETCH** transfers the values to the bound Q parameters	308
SQL ROLLBACK	**SQL ROLLBACK** discards all changes and concludes the transaction	314
SQL COMMIT	**SQL COMMIT** saves all changes and concludes the transaction	313
SQL UPDATE	**SQL UPDATE** expands the transaction to include the change of an existing row	310
SQL INSERT	**SQL INSERT** creates a new table row	312
SQL SELECT	**SQL SELECT** reads out a single value from a table and does not open any transaction	316

SQL BIND

SQL BIND links a Q parameter to a table column. The SQL commands **FETCH**, **UPDATE**, and **INSERT** evaluate this binding (assignment) during data transfer between the **result set** and the NC program.

An **SQL BIND** command without a table name or column name cancels the binding. At the latest, the binding is terminated at the end of the NC program or subprogram.



> Programming notes:
> - Program any number of bindings with **SQL BIND...**, before using the **FETCH**, **UPDATE**, or **INSERT** commands.
> - During the read and write operations, the control considers only those columns that you have specified by means of the **SELECT** command. If you specify columns without a binding in the **SELECT** command, then the control interrupts the read or write operation with an error message.

SQL BIND

- **Parameter no. for result**: Define Q parameter for binding to the table column
- **Database: column name**: Define table name and table column (separate with **.**)
  - **Table name**: Synonym or path with filename of the table
  - **Column name**: Name displayed in the table editor

Example: Binding Q parameters to table columns

```
11 SQL BIND Q881 "Tab_Example.Position_Nr"
12 SQL BIND Q882 "Tab_Example.Measure_X"
13 SQL BIND Q883 "Tab_Example.Measure_Y"
14 SQL BIND Q884 "Tab_Example.Measure_Z"
```

Example: Remove binding

```
91 SQL BIND Q881
92 SQL BIND Q882
93 SQL BIND Q883
94 SQL BIND Q884
```

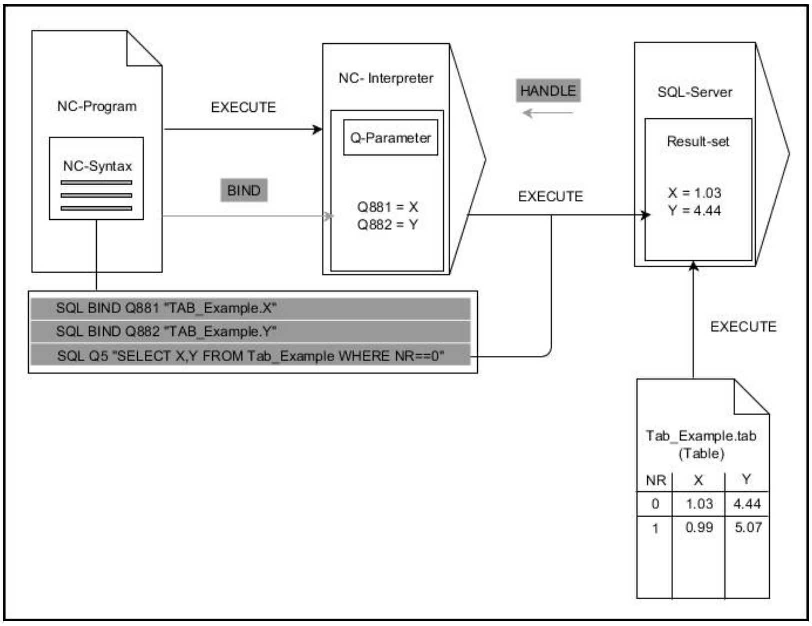
SQL EXECUTE

SQL EXECUTE can be used in conjunction with various SQL instructions.

The following SQL instructions are used in the SQL command **SQL EXECUTE**.

Instruction	Function
SELECT	Select data
CREATE SYNONYM	Create synonym (replace long path names with short names)
DROP SYNONYM	Delete synonym
CREATE TABLE	Generate a table
COPY TABLE	Copy table
RENAME TABLE	Rename table
DROP TABLE	Delete a table
INSERT	Insert table rows
UPDATE	Update table rows
DELETE	Delete table rows
ALTER TABLE	■ Add table columns using **ADD** ■ Delete table columns using **DROP**
RENAME COLUMN	Rename table columns

Example for the SQL EXECUTE command



Remarks:

- The gray arrows and associated syntax do not directly belong to the **SQL EXECUTE** command
- Black arrows and associated syntax indicate internal processes of **SQL EXECUTE**

SQL EXECUTE with the SQL instruction SELECT

The SQL server places the data in the **result set** row-by-row. The rows are numbered in ascending order, starting with 0. These row numbers (the **INDEX**) use the SQL commands **FETCH** and **UPDATE**.

SQL EXECUTE, in conjunction with the SQL instruction **SELECT**, selects the table values, transfers them to the **result set**, and always opens a transaction in the process. Unlike the SQL command **SQL SELECT**, the combination of **SQL EXECUTE** and the **SELECT** instruction allows multiple columns and rows to be selected at the same time.

In the function **SQL ... "SELECT...WHERE..."**, you can enter the search criteria. You thereby restrict the number of rows to be transferred. If you do not use this option, then all of the rows in the table are loaded.

In the function **SQL ... "SELECT...ORDER BY..."**, you can enter the ordering criterion. This entry consists of the column designation and the keyword **ASC** for ascending or **DESC** for descending order. If you do not use this option, then rows will be stored in a random order.

With the function **SQL ... "SELECT...FOR UPDATE"**, you can lock the selected rows for other applications. Other applications can continue to read these rows but are unable to change them. If you make changes to the table entries, then it is absolutely necessary to use this option.

Empty result set: If no rows meet the search criterion, then the SQL server returns a valid **HANDLE** without table entries.

SQL EXECUTE

- Define **Parameter number for result**
  - The return value serves as an identifying feature of a successfully opened transaction
  - The return value is used to control the read operation

  In the specified parameters, the control stores the **HANDLE** under which the read operation will subsequently occur. The **HANDLE** is valid until you confirm or reject the transaction.
  - **0**: Faulty read operation
  - Unequal to **0**: Return value of the **HANDLE**
- **Database: SQL instruction**: Program an SQL instruction
  - **SELECT**: Table columns to be transferred (separate multiple columns with **,**)
  - **FROM**: Synonym or absolute path of the table (path in single quotation marks)
  - **WHERE** (optional): Column names, condition, and comparison value (Q parameters after **:** in single quotation marks)
  - **ORDER BY** (optional): Column names and type of ordering (**ASC** for ascending and **DESC** for descending order)
  - **FOR UPDATE** (optional): To lock other processes from performing a write access to the selected rows

Conditions for WHERE entries

Condition	Programming
Equals	= ==
Not equal to	!= <>
Less than	<
Less than or equal to	<=
Greater than	>
Greater than or equal to	>=
empty	IS NULL
Not empty	IS NOT NULL
Linking multiple conditions:	
Logical AND	AND
Logical OR	OR

Example: selection of table rows

```
11 SQL BIND Q881 "Tab_Example.Position_Nr"
12 SQL BIND Q882 "Tab_Example.Measure_X"
13 SQL BIND Q883 "Tab_Example.Measure_Y"
14 SQL BIND Q884 "Tab_Example.Measure_Z"
. . .
20 SQL Q5 "SELECT Position_Nr,Measure_X,Measure_Y,
   Measure_Z FROM Tab_Example"
```

Example: Select table rows with the WHERE function

```
20 SQL Q5 "SELECT Position_Nr,Measure_X,Measure_Y,
   Measure_Z FROM Tab_Example WHERE
   Position_Nr<20"
```

Example: Select table rows with the WHERE function and Q parameter

```
20 SQL Q5 "SELECT Position_Nr,Measure_X,Measure_Y,
   Measure_Z FROM Tab_Example WHERE
   Position_Nr==:’Q11’"
```

Example: Define the table name with absolute path information

```
20 SQL Q5 "SELECT Position_Nr,Measure_X,Measure_Y,
   Measure_Z FROM ’V:\table\Tab_Example’ WHERE
   Position_Nr<20"
```

Example: Generate a table with CREATE TABLE

`0 BEGIN PGM SQL_CREATE_TAB MM`	
`1 SQL Q10 "CREATE SYNONYM NEW FOR 'TNC:\table \NewTab.TAB'"`	Create synonym
`2 SQL Q10 "CREATE TABLE NEW AS SELECT X,Y,Z FROM 'TNC:\prototype_for_NewTab.tab'"`	Create table
`3 END PGM SQL_CREATE_TAB MM`	

You can also define synonyms for tables that have not yet been generated.

The sequence of the columns in the created file corresponds to the sequence within the **AS SELECT** instruction.

Example: Generate a table with CREATE TABLE and QS

For the instructions within the SQL command, you can likewise use single or combined QS parameters.

If you check the content of a QS parameter in the additional status indicator (**QPARA** tab), then you will see only the first 30 characters and therefore not the complete content.

0 BEGIN PGM SQL_CREATE_TABLE_QS MM	
1 DECLARE STRING QS1 = "CREATE TABLE "	
2 DECLARE STRING QS2 = "'TNC:\nc_prog\demo\Doku \NewTab.t' "	
3 DECLARE STRING QS3 = "AS SELECT "	
4 DECLARE STRING QS4 = "DL,R,DR,L "	
5 DECLARE STRING QS5 = "FROM "	
6 DECLARE STRING QS6 = "'TNC:\table\tool.t'"	
7 QS7 = QS1 || QS2 || QS3 || QS4 || QS5 || QS6	
8 SQL Q1800 QS7	
9 END PGM SQL_CREATE_TABLE_QS MM	

Examples

The following examples do not result in a cohesive NC program. The NC blocks show only possible uses of the SQL command **SQL EXECUTE**.

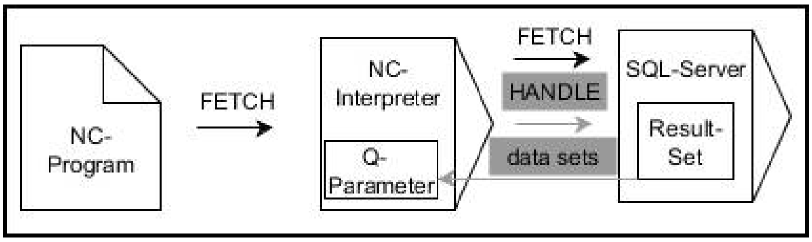
9 SQL Q1800 "CREATE SYNONYM my_table FOR 'TNC:\table\WMAT.TAB'"	Create synonym
9 SQL Q1800 "DROP SYNONYM my_table"	Delete synonym
9 SQL Q1800 "CREATE TABLE my_table (NO,WMAT)"	Create table with the rows NO and WMAT.
9 SQL Q1800 "COPY TABLE my_table TO 'TNC:\table \WMAT2.TAB'"	Copy table
9 SQL Q1800 "RENAME TABLE my_table TO 'TNC:\table \WMAT3.TAB'"	Rename table
9 SQL Q1800 "DROP TABLE my_table"	Delete the table
9 SQL Q1800 "INSERT INTO my_table VALUES (1,'ENAW',240)"	Insert table row
9 SQL Q1800 "DELETE FROM my_table WHERE NR==3"	Delete table row
9 SQL Q1800 "ALTER TABLE my_table ADD (WMAT2)"	Insert table rows
9 SQL Q1800 "ALTER TABLE my_table DROP (WMAT2)"	Delete table row
9 SQL Q1800 "RENAME COLUMN my_table (WMAT2) TO (WMAT3)"	Rename table column

SQL FETCH

SQL FETCH reads a row from the **result set**. The values of the individual cells are stored by the control in the bound Q parameters. The transaction is defined through the **HANDLE** to be specified, and the row is defined by the **INDEX**.

SQL FETCH takes all of the columns into consideration that contain the **SELECT** instruction (SQL command **SQL EXECUTE**).

Example for the SQL FETCH command



Remarks:

- The gray arrows and associated syntax do not directly belong to the **SQL FETCH** command
- Black arrows and associated syntax indicate internal processes of **SQL FETCH**

- Define **Parameter number for result** (return values for the control):
  - **0**: Successful read operation
  - **1**: Faulty read operation
- **Database: SQL access ID**: Define Q parameter for the **HANDLE** (for identifying the transaction)
- Define **Database: Index for SQL result** (row number within the **result set**)
  - Row number
  - Q parameter with the index
  - None defined: access to row 0

> The optional syntax elements **IGNORE UNBOUND** and **UNDEFINE MISSING** are intended for the machine manufacturer.

Example: Transfer row number in the Q parameter

```
11 SQL BIND Q881 "Tab_Example.Position_Nr"
12 SQL BIND Q882 "Tab_Example.Measure_X"
13 SQL BIND Q883 "Tab_Example.Measure_Y"
14 SQL BIND Q884 "Tab_Example.Measure_Z"
. . .
20 SQL Q5 "SELECT Position_Nr,Measure_X,Measure_Y,
   Measure_Z FROM Tab_Example"
. . .
30 SQL FETCH Q1 HANDLE Q5 INDEX+Q2
```

Example: Program the row number directly

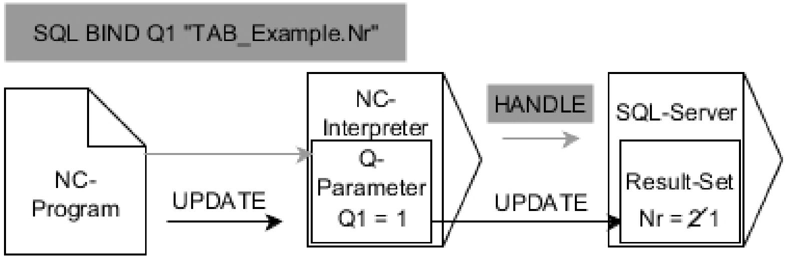
```
30 SQL FETCH Q1 HANDLE Q5 INDEX5
```

SQL UPDATE

SQL UPDATE changes a row in the **result set**. The new values of the individual cells are copied by the control from the bound Q parameters. The transaction is defined through the **HANDLE** to be specified, and the row is defined by the **INDEX**. The control completely overwrites the already existing rows in the **result set**.

SQL UPDATE takes all of the columns into consideration that contain the **SELECT** instruction (SQL command **SQL EXECUTE**).

Example for the SQL UPDATE command



The gray arrows and associated syntax do not directly belong to the **SQL UPDATE**
Black arrows and associated syntax show internal processes of **SQL UPDATE**

SQL UPDATE

- Define **Parameter number for result** (return values for the control):
  - **0**: Change was successful
  - **1**: Change failed
- **Database: SQL access ID**: Define Q parameter for the **HANDLE** (for identifying the transaction)
- Define **Database: Index for SQL result** (row number within the **result set**)
  - Row number
  - Q parameter with the index
  - None defined: access to row 0

When writing to tables, the control checks the lengths of the string parameters. If the entries exceed the length of the columns to be described, then the control outputs an error message.

Example: Transfer row number in the Q parameter

```
11 SQL BIND Q881 "TAB_EXAMPLE.Position_NR"
12 SQL BIND Q882 "TAB_EXAMPLE.Measure_X"
13 SQL BIND Q883 "TAB_EXAMPLE.Measure_Y"
14 SQL BIND Q884 "TAB_EXAMPLE.Measure_Z"
. . .
20 SQL Q5 "SELECT
    Position_NR,Measure_X,Measure_Y,Measure_Z FROM
    TAB_EXAMPLE"
. . .
30 SQL FETCH Q1 HANDLE Q5 INDEX+Q2
```

Example: Program the row number directly

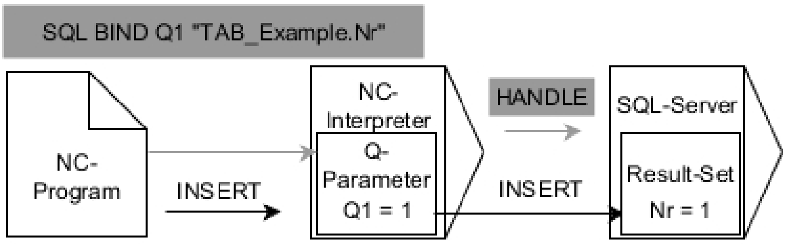
```
40 SQL UPDATE Q1 HANDLE Q5 INDEX5
```

SQL INSERT

SQL INSERT creates a new row in the **result set**. The values of the individual cells are copied by the control from the bound Q parameters. The transaction is defined via the **HANDLE** to be specified.

SQL INSERT takes all of the columns into consideration that contain the **SELECT** instruction (SQL command **SQL EXECUTE**). Table columns without a corresponding **SELECT** instruction (not contained in the query result) are described by the control with default values.

Example for the SQL INSERT command



Remarks:

- The gray arrows and associated syntax do not directly belong to the **SQL INSERT** command
- Black arrows and associated syntax indicate internal processes of **SQL INSERT**

- ▶ Define **Parameter number for result** (return values for the control):
  - **0**: Transaction successful
  - **1**: Transaction failed
- ▶ **Database: SQL access ID**: Define Q parameter for the **HANDLE** (for identifying the transaction)

> When writing to tables, the control checks the lengths of the string parameters. If the entries exceed the length of the columns to be described, then the control outputs an error message.

Example: Transfer row number in the Q parameter

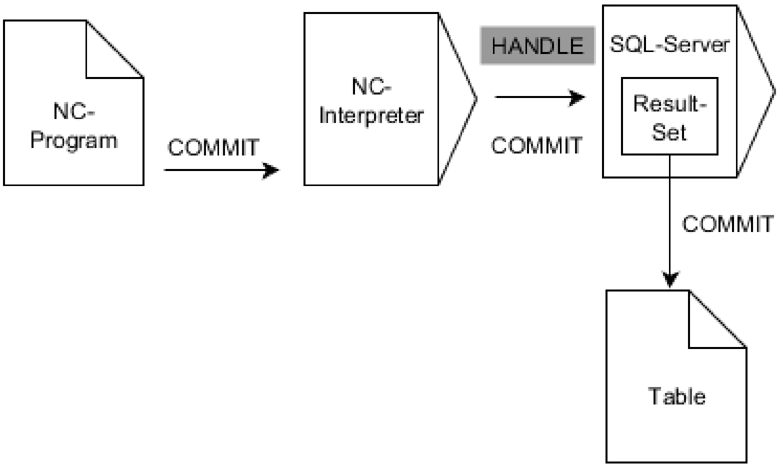
```
11 SQL BIND Q881 "Tab_Example.Position_Nr"
12 SQL BIND Q882 "Tab_Example.Measure_X"
13 SQL BIND Q883 "Tab_Example.Measure_Y"
14 SQL BIND Q884 "Tab_Example.Measure_Z"
. . .
20 SQL Q5 "SELECT Position_Nr,Measure_X,Measure_Y,
   Measure_Z FROM Tab_Example"
. . .
40 SQL INSERT Q1 HANDLE Q5
```

SQL COMMIT

SQL COMMIT simultaneously transfers all of the rows that have been changed and added in a transaction back into the table. The transaction is defined via the **HANDLE** to be specified. In this context, a lock that has been set with **SELECT...FOR UPDATE** resets the control.

The assigned **HANDLE** (operation) loses its validity.

Example for the SQL COMMIT command



Remarks:

- The gray arrows and associated syntax do not directly belong to the **SQL COMMIT** command
- Black arrows and associated syntax indicate internal processes of **SQL COMMIT**

- ▶ Define **Parameter number for result** (return values for the control):
  - **0**: Transaction successful
  - **1**: Transaction failed
- ▶ **Database: SQL access ID**: Define Q parameter for the **HANDLE** (for identifying the transaction)

Example

11 SQL BIND Q881 "Tab_Example.Position_Nr"	
12 SQL BIND Q882 "Tab_Example.Measure_X"	
13 SQL BIND Q883 "Tab_Example.Measure_Y"	
14 SQL BIND Q884 "Tab_Example.Measure_Z"	
. . .	
20 SQL Q5 "SELECT Position_Nr,Measure_X,Measure_Y, Measure_Z FROM Tab_Example"	
. . .	
30 SQL FETCH Q1 HANDLE Q5 INDEX+Q2	
. . .	
40 SQL UPDATE Q1 HANDLE Q5 INDEX+Q2	
. . .	
50 SQL COMMIT Q1 HANDLE Q5	

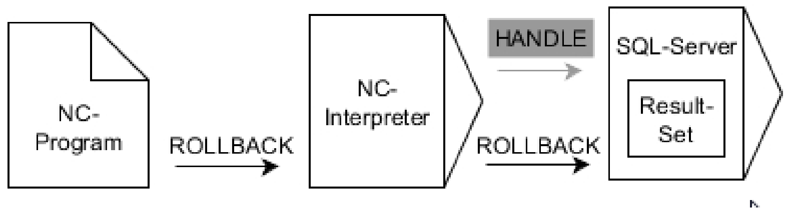
SQL ROLLBACK

SQL ROLLBACK discards all of the changes and additions of a transaction. The transaction is defined via the **HANDLE** to be specified.

The function of the SQL command **SQL ROLLBACK** depends on the **INDEX**:

- Without **INDEX**:
  - The control discards all changes and additions of the transaction
  - The control resets a lock set with **SELECT...FOR UPDATE**
  - The control completes the transaction (the **HANDLE** loses its validity)
- With **INDEX**:
  - Only the indexed row remains in the **result set** (the control removes all of the other rows)
  - The control discards any changes and additions that may have been made in the non-specified rows
  - The control locks only those rows indexed with **SELECT...FOR UPDATE** (the control resets all of the other locks)
  - The specified (indexed) row is then the new Row 0 of the **result set**
  - The control does **not** complete the transaction (the **HANDLE** keeps its validity)
  - The transaction must be completed manually with **SQL ROLLBACK** or **SQL COMMIT** at a later time

Example for the SQL ROLLBACK command



Remarks:

- The gray arrows and associated syntax do not directly belong to the **SQL ROLLBACK** command
- Black arrows and associated syntax indicate internal processes of **SQL ROLLBACK**

SQL ROLLBACK

- Define **Parameter number for result** (return values for the control):
  - **0**: Transaction successful
  - **1**: Transaction failed
- **Database: SQL access ID**: Define Q parameter for the **HANDLE** (for identifying the transaction)
- Define **Database: Index for SQL result** (row that remains in the **result set**)
  - Row number
  - Q parameter with the index

Example

11 SQL BIND Q881 "Tab_Example.Position_Nr"	
12 SQL BIND Q882 "Tab_Example.Measure_X"	
13 SQL BIND Q883 "Tab_Example.Measure_Y"	
14 SQL BIND Q884 "Tab_Example.Measure_Z"	
. . .	
20 SQL Q5 "SELECT Position_Nr,Measure_X,Measure_Y, Measure_Z FROM Tab_Example"	
. . .	
30 SQL FETCH Q1 HANDLE Q5 INDEX+Q2	
. . .	
50 SQL ROLLBACK Q1 HANDLE Q5	

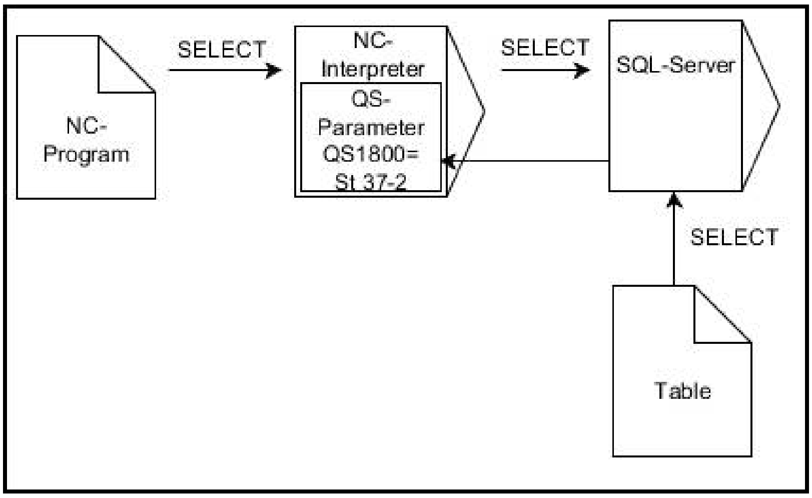
SQL SELECT

SQL SELECT reads a single value from a table and saves the result in the defined Q parameter.

> You can select multiple values or multiple columns using the SQL command **SQL EXECUTE** and the **SELECT** instruction.
> **Further information:** "SQL EXECUTE", Page 304

With **SQL SELECT**, there is neither a transaction nor a binding between the table column and Q parameter. The control does not consider any bindings that may exist to the specified column. The control copies the read value only into the parameter specified for the result.

Example for the SQL SELECT command



Remark:

- Black arrows and associated syntax show internal processes of **SQL SELECT**

- Define **Parameter number for result** (Q parameter for saving the value)
- **Database: SQL command text**: Program the SQL instruction
  - **SELECT**: Table column of the value to be transferred
  - **FROM**: Synonym or absolute path of the table (path in single quotation marks)
  - **WHERE**: Column designation, condition, and comparison value (Q parameter after **:** in single quotation marks)

Example: Read and save a value

```
20 SQL SELECT Q5 "SELECT Mess_X FROM Tab_Example
   WHERE Position_NR==3"
```

Comparison

The results of the following NC programs are identical.

0 BEGIN PGM SQL_READ_WMAT MM	
1 SQL Q1800 "CREATE SYNONYM my_table FOR 'TNC:\table\WMAT.TAB'"	Create synonym
2 SQL BIND QS1800 "my_table.WMAT"	Bind QS parameters
3 SQL QL1 "SELECT WMAT FROM my_table WHERE NR==3"	Define search
...	

...	
3 SQL SELECT QS1800 "SELECT WMAT FROM my_table WHERE NR==3"	Read and save a value
...	

For the instructions within the SQL command, you can likewise use single or combined QS parameters.

If you check the content of a QS parameter in the additional status indicator (**QPARA** tab), then you will see only the first 30 characters and therefore not the complete content.

...	
3 DECLARE STRING QS1 = "SELECT "	
4 DECLARE STRING QS2 = "WMAT "	
5 DECLARE STRING QS3 = "FROM "	
6 DECLARE STRING QS4 = "my_table "	
7 DECLARE STRING QS5 = "WHERE "	
8 DECLARE STRING QS6 = "NR==3"	
9 QS7 = QS1 || QS2 || QS3 || QS4 || QS5 || QS6	
10 SQL SELECT QL1 QS7	
11 ...	

Examples

In the following example, the defined material is read from the table (**WMAT.TAB**) and is stored as a text in a QS parameter. The following example shows a possible application and the necessary program steps.

> You can use the **FN 16** function, for example, in order to reuse QS parameters in your own log files.
> **Further information:** "Basics", Page 285

Example: Use a synonym

0 BEGIN PGM SQL_READ_WMAT MM	
1 SQL Q1800 "CREATE SYNONYM my_table FOR 'TNC:\table\WMAT.TAB'"	Create synonym
2 SQL BIND QS1800 "my_table.WMAT"	Bind QS parameters
3 SQL QL1 "SELECT WMAT FROM my_table WHERE NO==3"	Define search
4 SQL FETCH Q1900 HANDLE QL1	Execute search
5 SQL ROLLBACK Q1900 HANDLE QL1	Complete transaction
6 SQL BIND QS1800	Remove parameter binding
7 SQL Q1 "DROP SYNONYM my_table"	Delete synonym
8 END PGM SQL_READ_WMAT MM	

Step	Explanation
1 Create synonym	Assign a synonym to a path (replace long paths with short names) ■ The path **TNC:\table\WMAT.TAB** is always placed in single quotes ■ The selected synonym is **my_table**
2 Bind QS parameters	Bind a QS parameter to a table column ■ **QS1800** is freely available in NC programs ■ The synonym replaces the entry of the complete path ■ The defined column from the table is called **WMAT**
3 Define search	A search definition contains the entry of the transfer value ■ The **QL1** local parameter (freely selectable) serves to identify the transaction (multiple transactions are possible simultaneously) ■ The synonym defines the table ■ The **WMAT** entry defines the table column of the read operation ■ The entries **NR** and **==3** define the table rows of the read operation ■ Selected table columns and rows define the cells of the read operation

4 Execute search	The control performs the read operation
	■ **SQL FETCH** copies the values from the **result set** into the bound Q or QS parameter
	  ■ **0** successful read operation
	  ■ **1** faulty read operation
	■ The **HANDLE QL1** syntax is the transaction designated by the **QL1** parameter
	■ The parameter **Q1900** is a return value for checking whether the data have been read

5 Complete transaction	The transaction is concluded and the used resources are released

Step	Explanation
6 Remove binding	The binding between table columns and QS parameters is removed (release of necessary resources)
7 Delete synonym	The synonym is deleted again (release of necessary resources)

> Synonyms are an alternative only to the required absolute paths. Relative path entries are not possible.

The following NC program shows the entry of an absolute path.

Example: Use an absolute path

0 BEGIN PGM SQL_READ_WMAT_2 MM	
1 SQL BIND QS 1800 "'TNC:\table\WMAT.TAB'.WMAT"	Bind QS parameters
2 SQL QL1 "SELECT WMAT FROM 'TNC:\table\WMAT.TAB' WHERE NR ==3"	Define search
3 SQL FETCH Q1900 HANDLE QL1	Execute search
4 SQL ROLLBACK Q1900 HANDLE QL1	Complete transaction
5 SQL BIND QS 1800	Remove parameter binding
6 END PGM SQL_READ_WMAT_2 MM	

9.10 Entering formulas directly

Entering formulas

Using soft keys, you can enter mathematical formulas containing multiple calculation operations directly into the NC program.

Q

- Select Q-parameter functions

FORMULA

- Press the **FORMULA** soft key
- Select **Q**, **QL**, or **QR**

The control displays the following soft keys in several soft-key rows:

Soft key	Linking function
+	**Addition** e.g., **Q10 = Q1 + Q5**
-	**Subtraction** e.g., **Q25 = Q7 - Q108**
*	**Multiplication** e.g., **Q12 = 5 * Q5**
/	**Division** e.g., **Q25 = Q1 / Q2**
(	**Opening parenthesis** e.g., **Q12 = Q1 * (Q2 + Q3)**
)	**Closing parenthesis** e.g., **Q12 = Q1 * (Q2 + Q3)**
SQ	**Square the value** e.g., **Q15 = SQ 5**
SQRT	**Calculate square root** e.g., **Q22 = SQRT 25**
SIN	**Sine of an angle** e.g., **Q44 = SIN 45**
COS	**Cosine of an angle** e.g., **Q45 = COS 45**
TAN	**Tangent of an angle** e.g., **Q46 = TAN 45**
ASIN	**Arc sine** Inverse function of the sine; determine the angle from the ratio of the opposite side to the hypotenuse e.g., **Q10 = ASIN 0.75**
ACOS	**Arc cosine** Inverse function of the cosine; determine the angle from the ratio of the adjacent side to the hypotenuse e.g., **Q11 = ACOS Q40**

Soft key	Linking function
ATAN	**Arc tangent** Inverse function of the tangent; determine the angle from the ratio of the opposite side to the adjacent side e.g., **Q12 = ATAN Q50**
^	**Powers of values** e.g., **Q15 = 3^3**
PI	**Constant PI (3,14159)** e.g., **Q15 = PI**
LN	**Calculate the natural logarithm of a number** Base 2.7183 e.g., **Q15 = LN Q11**
LOG	**Logarithm of a number, Base 10** e.g., **Q33 = LOG Q22**
EXP	**Exponential function, 2.7183 to the power of n** e.g., **Q1 = EXP Q12**
NEG	**Negate values (multiply by -1)** e.g., **Q2 = NEG Q1**
INT	**Remove digits after the decimal point** Calculate an integer e.g., **Q3 = INT Q42**
ABS	**Absolute value of a number** e.g., **Q4 = ABS Q22**
FRAC	**Remove digits before the decimal point** Calculate a fraction e.g., **Q5 = FRAC Q23**
SGN	**Check algebraic sign of a number** e g., **Q12 = SGN Q50** If return value Q12 = 0, then Q50 = 0 If return value Q12 = 1, then Q50 > 0 If return value Q12 = -1, then Q50 < 0
%	**Calculate modulo value (division remainder)** e.g., **Q12 = 400 % 360** Result: Q12 = 40

The **INT** function does not round off—it simply truncates the decimal places.

Further information: "Example: Rounding a value", Page 344

Rules for formulas

Mathematical formulas are programmed according to the following rules:

Higher-level operations are performed first

Example

```
12 Q1 = 5 * 3 + 2 * 10 = 35
```

1 Calculation 5 * 3 = 15
2 Calculation 2 * 10 = 20
3 Calculation 15 + 20 = 35

or

Example

```
13 Q2 = SQ 10 - 3^3 = 73
```

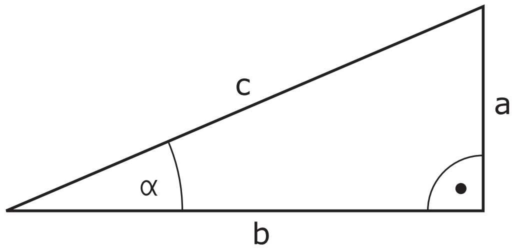
1 Calculation step 10 squared = 100
2 Calculation step 3 to the third power = 27
3 Calculation 100 – 27 = 73

Distributive law

Law of distribution with parentheses calculation
a * (b + c) = a * b + a * c

Example of entry

Calculate an angle with the arc tangent from the opposite side (Q12) and adjacent side (Q13); assign result to Q25.



Q FORMULA

- Select the formula entry function: Press the **Q** key and the **FORMULA** soft key, or use the shortcut

Q

- Press the **Q** key on the alphanumeric keyboard

PARAMETER NUMBER FOR RESULT?

ENT

- Enter **25** (parameter number) and press the **ENT** key

▷ ATAN

- Shift the soft-key row and select the arc tangent function

◁ (

- Advance through the soft key menu and press the **OPENING PARENTHESIS** soft key

Q

- Enter **12** (the parameter number)

/

- Select division

Q

- Enter **13** (the parameter number)

) END

- Close parentheses and conclude formula entry

Example

```
37 Q25 = ATAN (Q12/Q13)
```

9.11 String parameters

String processing functions

You can use the **QS** parameters to create variable character strings. You can output such character strings for example through the **FN 16:F-PRINT** function to create variable logs.

You can assign a linear sequence of characters (letters, numbers, special characters and spaces) up to a length of 255 characters to a string parameter. You can also check and process the assigned or imported values using the functions described below. As in Q parameter programming, you can use a total of 2000 QS parameters.

Further information: "Principle and overview of functions", Page 264

The **STRING FORMULA** and **FORMULA** Q parameter functions contain various functions for processing the string parameters.

Soft key	Functions of the STRING FORMULA	Page
STRING	Assigning string parameters	325
CFGREAD	Read out machine parameter	334
	Chain-linking string parameters	325
TOCHAR	Converting a numerical value to a string parameter	327
SUBSTR	Copy a substring from a string parameter	328
SYSSTR	Read system data	329

Soft key	Formula string functions	Page
TONUMB	Converting a string parameter to a numerical value	330
INSTR	Checking a string parameter	331
STRLEN	Finding the length of a string parameter	332
STRCOMP	Compare alphabetic priority	333

When you use the **STRING FORMULA** function, the result of the performed arithmetic operation is always a string. When you use the **FORMULA** function, the result of the performed arithmetic operation is always a numeric value.

Assign string parameters

Before using string variables, you must first assign the variables. Use the **DECLARE STRING** command to do so.

SPEC FCT

- Press the **SPEC FCT** key

PROGRAM FUNCTIONS

- Press the **PROGRAM FUNCTIONS** soft key

STRING FUNCTIONS

- Press the **STRING FUNCTIONS** soft key

DECLARE STRING

- Press the **DECLARE STRING** soft key

Example

```
37 DECLARE STRING QS10 = "Workpiece"
```

Chain-linking string parameters

With the concatenation operator (string parameter **||** string parameter) you can make a chain of two or more string parameters.

SPEC FCT

- Press the **SPEC FCT** key

PROGRAM FUNCTIONS

- Press the **PROGRAM FUNCTIONS** soft key

STRING FUNCTIONS

- Press the **STRING FUNCTIONS** soft key

STRING FORMULA

ENT

- Press the **STRING FORMULA** soft key
- Enter the number of the string parameter in which the control is to save the concatenated string. Confirm with the **ENT** key.
- Enter the number of the string parameter in which the **first** substring is saved. Confirm with the **ENT** key
- > The control shows the concatenation symbol **||** an.
- Press the **ENT** key
- Enter the number of the string parameter in which the **second** substring is saved. Confirm with the **ENT** key
- Repeat the process until you have selected all the required substrings. Conclude with the **END** key

Example: QS10 is to include the complete text of QS12, QS13 and QS14

```
37 QS10 =  QS12 || QS13 || QS14
```

Parameter contents:

- **QS12**: **Workpiece**
- **QS13**: **Status:**
- **QS14**: **Scrap**
- **QS10**: **Workpiece Status: Scrap**

Converting a numerical value to a string parameter

With the **TOCHAR** function, the control converts a numerical value into a string parameter. This enables you to chain numerical values with string variables.

SPEC FCT
- Show the soft-key row with special functions

PROGRAM FUNCTIONS
- Open the function menu

STRING FUNCTIONS
- Press the String functions soft key

STRING FORMULA
- Press the **STRING FORMULA** soft key

TOCHAR
- Select the function for converting a numerical value to a string parameter
- Enter the number or the desired Q parameter to be converted by the control, and confirm with the **ENT** key
- If desired, enter the number of digits after the decimal point that the control should convert, and confirm with the **ENT** key
- Close the parenthetical expression with the **ENT** key and confirm your entry with the **END** key

Example: Convert parameter Q50 to string parameter QS11, use 3 decimal places

```
37 QS11 = TOCHAR ( DAT+Q50 DECIMALS3 )
```

Copying a substring from a string parameter

The **SUBSTR** function copies a definable range from a string parameter.

SPEC FCT

- Show the soft-key row with special functions

PROGRAM FUNCTIONS

- Open the function menu

STRING FUNCTIONS

- Press the String functions soft key

STRING FORMULA

- Press the **STRING FORMULA** soft key
- Enter the number of the string parameter in which the control is to save the character string. Confirm with the **ENT** key.

SUBSTR

- Select the function for cutting out a substring
- Enter the number of the QS parameter from which the substring is to be copied. Confirm with the **ENT** key
- Enter the number of the place starting from which to copy the substring, and confirm with the **ENT** key
- Enter the number of characters to be copied, and confirm with the **ENT** key
- Close the parenthetical expression with the **ENT** key and confirm your entry with the **END** key

> The first character of a text string starts internally at the 0-position

Example: A four-character substring (LEN4) is read from the string parameter QS10 beginning with the third character (BEG2)

```
37 QS13 = SUBSTR ( SRC_QS10 BEG2 LEN4 )
```

Reading system data

With the function **SYSSTR** you can read system data and store them in string parameters. You select the system data through a group number (ID) and a number.

Entering IDX and DAT is not required.

Group name, ID no.	Number	Meaning
Program information, 10010	1	Path of the current main program or pallet program
	2	Path of the NC program shown in the block display
	3	Path of the cycle selected with **CYCL DEF 12 PGM CALL**
	10	Path of the NC program selected with **SEL PGM**
Channel data, 10025	1	Channel name
Values programmed in the tool call, 10060	1	Tool name
Kinematics, 10290	10	Kinematics programmed in the last **FUNCTION MODE** block
Current system time, 10321	1 - 16	▪ 1: DD.MM.YYYY hh:mm:ss ▪ 2 and 16: DD.MM.YYYY hh:mm ▪ 3: DD.MM.YY hh:mm ▪ 4: YYYY-MM-DD hh:mm:ss ▪ 5 and 6: YYYY-MM-DD hh:mm ▪ 7: YY-MM-DD hh:mm ▪ 8 and 9: DD.MM.YYYY ▪ 10: DD.MM.YY ▪ 11: YYYY-MM-DD ▪ 12: YY-MM-DD ▪ 13 and 14: hh:mm:ss ▪ 15: hh:mm
Touch-probe data, 10350	50	Probe type of the active touch probe TS
	70	Probe type of the active touch probe TT
	73	Key name of the active touch probe TT from MP **activeTT**
Data for pallet machining, 10510	1	Pallet name
	2	Path of the selected pallet table
NC software version, 10630	10	Version identifier of the NC software version
Information for unbalance cycle, 10855	1	Path of the unbalance calibration table belonging to the active kinematics
Tool data, 10950	1	Tool name
	2	DOC entry of the tool
	3	AFC control setting
	4	Tool-carrier kinematics

Converting a string parameter to a numerical value

The **TONUMB** function converts a string parameter to a numerical value. The value to be converted should be only numerical.

> The QS parameter to be converted must contain only one numerical value. Otherwise, the Control will output an error message.

Q

- Select Q-parameter functions

FORMULA

- Press the **FORMULA** soft key
- Enter the number of the string parameter in which the control is to save the numerical value. Confirm with the **ENT** key.

◁

- Shift the soft-key row

TONUMB

- Select the function for converting a string parameter to a numerical value
- Enter the number of the QS parameter to be converted by the control, and confirm with the **ENT** key
- Close the parenthetical expression with the **ENT** key and confirm your entry with the **END** key

Example: Convert string parameter QS11 to a numerical parameter Q82

```
37 Q82 = TONUMB ( SRC_QS11 )
```

Testing a string parameter

The **INSTR** function checks whether a string parameter is contained in another string parameter.

Q
- Select Q-parameter functions

FORMULA
- Press the **FORMULA** soft key
- Enter the number of the Q parameter for the result and confirm with the **ENT** key
- > The control saves the place at which the text to be searched for begins. It is saved in the parameter.

◁
- Shift the soft-key row

INSTR
- Select the function for checking a string parameter
- Enter the number of the QS parameter in which the text to be searched for is saved. Confirm with the **ENT** key
- Enter the number of the QS parameter to be searched for by the control, and confirm with the **ENT** key
- Enter the number of the place at which the control is to start search the substring, and confirm with the **ENT** key.
- Close the parenthetical expression with the **ENT** key and confirm your entry with the **END** key

> i
> The first character of a text string starts internally at the 0-position
>
> If the control cannot find the required substring, it will save the total length of the string to be searched (counting starts at 1) in the result parameter.
>
> If the substring to be searched for appears multiple times, then the control returns the first place at which it finds the substring.

Example: Search through QS10 for the text saved in parameter QS13. Begin the search at the third place.

```
37 Q50 = INSTR ( SRC_QS10 SEA_QS13 BEG2 )
```

Finding the length of a string parameter

The **STRLEN** function returns the length of the text saved in a selectable string parameter.

Q

- Select Q parameter function

FORMULA

- Press the **FORMULA** soft key
- Enter the number of the Q parameter in which the control is to save the ascertained string length. Confirm with the **ENT** key.

◁

- Shift the soft-key row

STRLEN

- Select the function for finding the text length of a string parameter
- Enter the number of the QS parameter from which the control is to ascertain the length, and confirm with the **ENT** key
- Close the parenthetical expression with the **ENT** key and confirm your entry with the **END** key

Example: Find the length of QS15

```
37 Q52 = STRLEN ( SRC_QS15 )
```

If the selected string parameter is not defined the control returns the result **-1**.

Comparing alphabetic priority

The **STRCOMP** function compares string parameters for alphabetic priority.

Q

- Select Q parameter function

FORMULA

- Press the **FORMULA** soft key
- Enter the number of the Q parameter in which the control is to save the result of comparison, and confirm with the **ENT** key.

◁

- Shift the soft-key row

STRCOMP

- Select the function for comparing string parameters
- Enter the number of the first QS parameter that the control is to compare, and confirm with the **ENT** key
- Enter the number of the second QS parameter that the control is to compare, and confirm with the **ENT** key
- Close the parenthetical expression with the **ENT** key and confirm your entry with the **END** key

The control returns the following results:

- **0**: The compared QS parameters are identical
- **-1**: The first QS parameter **precedes** the second QS parameter alphabetically
- **+1:** The first QS parameter **follows** the second QS parameter alphabetically

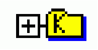
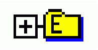
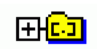
Example: QS12 and QS14 are compared for alphabetic priority

```
37 Q52 = STRCOMP  ( SRC_QS12 SEA_QS14 )
```

Reading out machine parameters

With the **CFGREAD** function, you can read out machine parameters of the control as numerical values or as strings. The read-out values are always output in metric units of measure.

In order to read out a machine parameter, you must use the control's configuration editor to determine the parameter name, parameter object, and, if they have been assigned, the group name and index:

Icon	Type	Meaning	Example
	**Key**	Group name of the machine parameter (if available)	CH_NC
	**Entity**	Parameter object (name begins with **Cfg**...)	**CfgGeoCycle**
	Attribute	Name of the machine parameter	**displaySpindleErr**
	**Index**	List index of a machine parameter (if available)	[0]

If you are in the configuration editor for the user parameters, you can change the display of the existing parameters. In the default setting, the parameters are displayed with short, explanatory texts.

Further information: User's Manual for Setup, Testing and Running NC Programs

Each time you want to interrogate a machine parameter with the **CFGREAD** function, you must first define a QS parameter with attribute, entity and key.

The following parameters are read in the CFGREAD function’s dialog:

- **KEY_QS**: Group name (key) of the machine parameter
- **TAG_QS**: Object name (entity) of the machine parameter
- **ATR_QS**: Name (attribute) of the machine parameter
- **IDX**: Index of the machine parameter

Reading a string of a machine parameter

In order to store the content of a machine parameter as a string in a QS parameter:

Q

- Press the **Q** key.

STRING FORMULA

- Press the **STRING FORMULA** soft key
- Enter the number of the string parameter in which the control is to save the machine parameter
- Press the **ENT** key
- Select the **CFGREAD** function
- Enter the numbers of the string parameters for key, entity, and attribute
- Press the **ENT** key
- Enter the number for the index, or skip the dialog with N**NO ENT**, whichever applies
- Close the parenthesized expression with the **ENT** key
- Press the **END** key to conclude entry

Example: Read as a string the axis designation of the fourth axis

Parameter settings in the configuration editor

DisplaySettings

CfgDisplayData

axisDisplayOrder

[0] to [5]

Example

14 QS11 = ""	Assign string parameter for key
15 QS12 = "CfgDisplaydata"	Assign string parameter for entity
16 QS13 = "axisDisplay"	Assign string parameter for parameter name
17 QS1 = CFGREAD(KEY_QS11 TAG_QS12 ATR_QS13 IDX3)	Read out machine parameter

Reading a numerical value of a machine parameter

Store the value of a machine parameter as a numerical value in a Q parameter:

Q

- Select Q parameter function

FORMULA

- Press the **FORMULA** soft key
- Enter the number of the Q parameter in which the control is to save the machine parameter
- Press the **ENT** key
- Select the **CFGREAD** function
- Enter the numbers of the string parameters for key, entity, and attribute
- Press the **ENT** key
- Enter the number for the index, or skip the dialog with N**NO ENT**, whichever applies
- Close the parenthesized expression with the **ENT** key
- Press the **END** key to conclude entry

Example: Read overlap factor as Q parameter

Parameter settings in the configuration editor

ChannelSettings
CH_NC
CfgGeoCycle
pocketOverlap

Example

14 QS11 = "CH_NC"	Assign string parameter for key
15 QS12 = "CfgGeoCycle"	Assign string parameter for entity
16 QS13 = "pocketOverlap"	Assign string parameter for parameter name
17 Q50 = CFGREAD(KEY_QS11 TAG_QS12 ATR_QS13)	Read out machine parameter

9.12 Preassigned Q parameters

The Q parameters Q100 to Q199 are assigned values by the control. The following types of information are assigned to the Q parameters:

- Values from the PLC
- Tool and spindle data
- Data on operating status
- Results of measurements from touch probe cycles etc.

The control saves the preassigned Q parameters Q108, Q114, and Q115 to Q117 in the unit of measure used by the active NC program.

NOTICE

Danger of collision!

HEIDENHAIN cycles, manufacturer cycles and third-party functions use Q parameters. You can also program Q parameters within NC programs. If, when using Q parameters, the recommended Q parameter ranges are not used exclusively, then this can lead to overlapping (reciprocal effects) and thus cause undesired behavior. Danger of collision during machining!

- Only use Q parameter ranges recommended by HEIDENHAIN.
- Comply with the documentation from HEIDENHAIN, the machine tool builder, and suppliers.
- Check the machining sequence using a graphic simulation

You must not use preassigned Q parameters (QS parameters) between **Q100** and **Q199** (**QS100** and **QS199**) as calculation parameters in the NC programs.

Values from the PLC: Q100 to Q107

The control assigns values from the PLC to parameters Q100 to Q107 in an NC program.

Active tool radius: Q108

The active value of the tool radius is assigned to Q108. Q108 is calculated from:

- Tool radius R (tool table or **TOOL DEF** block)
- Delta value DR from the tool table
- Delta value DR from the NC program (compensation table or **TOOL CALL** block)

The control remembers the current tool radius even if the power is interrupted.

Tool axis: Q109

The value of Q109 depends on the current tool axis:

Parameter	Tool axis
Q109 = –1	No tool axis defined
Q109 = 0	X axis
Q109 = 1	Y axis
Q109 = 2	Z axis
Q109 = 6	U axis
Q109 = 7	V axis
Q109 = 8	W axis

Spindle status: Q110

The value of the parameter Q110 depends on the M function last programmed for the spindle.

Parameter	M function
Q110 = –1	No spindle status defined
Q110 = 0	M3: Spindle ON, clockwise
Q110 = 1	M4: Spindle ON, counterclockwise
Q110 = 2	M5 after M3
Q110 = 3	M5 after M4

Coolant on/off: Q111

Parameter	M function
Q111 = 1	M8: Coolant ON
Q111 = 0	M9: Coolant OFF

Overlap factor: Q112

The control assigns Q112 to the overlap factor for pocket milling.

Unit of measurement for dimensions in the NC program: Q113

During nesting the **PGM CALL**, the value of the parameter Q113 depends on the dimensional data of the NC program from which the other NC programs are called.

Parameter	Dimensional data of the main program
Q113 = 0	Metric system (mm)
Q113 = 1	Imperial system (inch)

Tool length: Q114

The current value for the tool length is assigned to Q114.

> The Control remembers the current tool length even if the power is interrupted.

Coordinates after probing during program run

The parameters Q115 to Q119 contain the coordinates of the spindle position at the moment of contact during programmed measurement with the 3-D touch probe. The coordinates are referenced to the preset that is active in **Manual operation** mode.

The length of the stylus and the radius of the ball tip are not compensated in these coordinates.

Parameter	Coordinate axis
Q115	X axis
Q116	Y axis
Q117	Z axis
Q118	4th axis Machine-dependent
Q119	5th axis Machine-dependent

Deviation between actual and nominal value during automatic tool measurement; for example, with the TT 160

Parameter	Deviation of actual from nominal value
Q115	Tool length
Q116	Tool radius

Tilting the working plane with workpiece angles: Coordinates calculated by the control for rotary axes

Parameter	Coordinates
Q120	A axis
Q121	B axis
Q122	C axis

Measurement results from touch probe cycles

Further information: Cycle Programming User's Manual

Parameters	Measured actual values
Q150	Angle of a straight line
Q151	Center in reference axis
Q152	Center in minor axis
Q153	Diameter
Q154	Pocket length
Q155	Pocket width
Q156	Length of the axis selected in the cycle
Q157	Position of the centerline
Q158	Angle in the A axis
Q159	Angle in the B axis
Q160	Coordinate of the axis selected in the cycle

Parameters	Measured deviation
Q161	Center in reference axis
Q162	Center in minor axis
Q163	Diameter
Q164	Pocket length
Q165	Pocket width
Q166	Measured length
Q167	Position of the centerline

Parameters	Determined space angle
Q170	Rotation about the A axis
Q171	Rotation about the B axis
Q172	Rotation about the C axis

Parameters	Workpiece status
Q180	Good
Q181	Rework
Q182	Scrap

Parameters	Tool measurement with the BLUM laser
Q190	Reserved
Q191	Reserved
Q192	Reserved
Q193	Reserved

Parameters	Reserved for internal use
Q195	Marker for cycles
Q196	Marker for cycles
Q197	Marker for cycles (machining patterns)
Q198	Number of the last active measuring cycle

Parameter value	Status of tool measurement with TT
Q199 = 0.0	Tool is within the tolerance.
Q199 = 1.0	Tool is worn (LTOL/RTOL is exceeded)
Q199 = 2.0	Tool is broken (LBREAK/RBREAK is exceeded)

Measurement results from touch probe cycles 14xx

Parameters	Measured actual values
Q950	1st position in the reference axis
Q951	1st position in the minor axis
Q952	1st position in the tool axis
Q953	2nd position in the reference axis
Q954	2nd position in the minor axis
Q955	2nd position in the tool axis
Q956	3rd position in the reference axis
Q957	3rd position in the minor axis
Q958	3rd position in the tool axis
Q961	Spatial angle SPA in the WPL-CS
Q962	Spatial angle SPB in the WPL-CS
Q963	Spatial angle SPC in the WPL-CS
Q964	Angle of rotation in the I-CS
Q965	Angle of rotation in the coordinate system of the rotary table
Q966	First diameter
Q967	Second diameter

Parameters	Measured deviations
Q980	1st position in the reference axis
Q981	1st position in the minor axis
Q982	1st position in the tool axis
Q983	2nd position in the reference axis
Q984	2nd position in the minor axis
Q985	2nd position in the tool axis
Q986	3rd position in the reference axis
Q987	3rd position in the minor axis
Q988	3rd position in the tool axis
Q994	Angle in the I-CS
Q995	Angle in the coordinate system of the rotary table
Q996	First diameter
Q997	Second diameter

Parameter value	Workpiece status
Q183 = -1	Not defined
Q183 = 0	Pass
Q183 = 1	Rework
Q183 = 2	Scrap

Checking the setup situation: Q601

The value of the parameter Q601 indicates the status of the camera-based monitoring of the VSC setup situation.

Parameter value	Status
Q601 = 1	No error
Q601 = 2	Error
Q601 = 3	No monitoring area defined or not enough reference images
Q601 = 10	Internal error (no signal, camera error, etc.)

9.13 Programming examples

Example: Rounding a value

The **INT** function truncates the decimal places.

In order for the control to round correctly, rather than simply truncating the decimal places, add the value 0.5 to a positive number. For a negative number you must subtract 0.5.

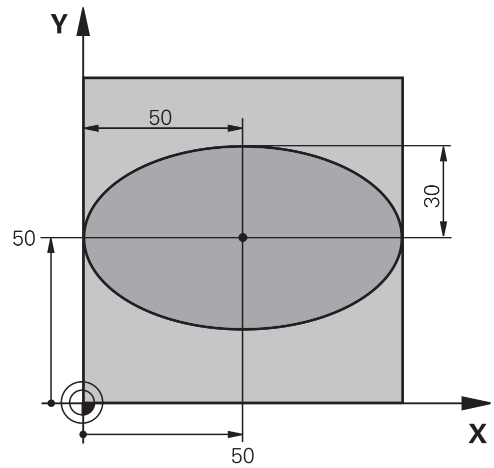
The control uses the **SGN** function to detect whether a number is positive or negative.

0 BEGIN PGM ROUND MM	
1 FN 0: Q1 = +34.789	First number to be rounded
2 FN 0: Q2 = +34.345	Second number to be rounded
3 FN 0: Q3 = -34.432	Third number to be rounded
4 ;	
5 Q11 = INT (Q1 + 0.5 * SGN Q1)	Add the value 0.5 to Q1, then truncate the decimal places
6 Q12 = INT (Q2 + 0.5 * SGN Q2)	Add the value 0.5 to Q2, then truncate the decimal places
7 Q13 = INT (Q3 + 0.5 * SGN Q3)	Subtract the value 0.5 from Q3, then truncate the decimal places
8 END PGM ROUND MM	

Example: Ellipse

Program run

- The contour of the ellipse is approximated by many short lines (defined in Q7). The more calculation steps you define for the lines, the smoother the curve becomes.
- The milling direction is determined with the starting angle and end angle in the plane:
  Machining direction is clockwise:
  Starting angle > end angle
  Machining direction is counterclockwise:
  Starting angle < end angle
- The tool radius is not taken into account



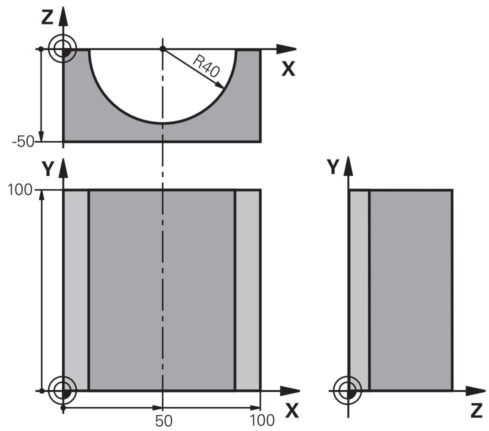
0 BEGIN PGM ELLIPSE MM	
1 FN 0: Q1 = +50	Center in X axis
2 FN 0: Q2 = +50	Center in Y axis
3 FN 0: Q3 = +50	Semiaxis in X
4 FN 0: Q4 = +30	Semiaxis in Y
5 FN 0: Q5 = +0	Starting angle in the plane
6 FN 0: Q6 = +360	End angle in the plane
7 FN 0: Q7 = +40	Number of calculation steps
8 FN 0: Q8 = +0	Rotational position of the ellipse
9 FN 0: Q9 = +5	Milling depth
10 FN 0: Q10 = +100	Feed rate for plunging
11 FN 0: Q11 = +350	Feed rate for milling
12 FN 0: Q12 = +2	Set-up clearance for pre-positioning
13 BLK FORM 0.1 Z X+0 Y+0 Z-20	Workpiece blank definition
14 BLK FORM 0.2 X+100 Y100 Z+0	
15 TOOL CALL 1 Z S4000	Tool call
16 L Z+250 R0 FMAX	Retract the tool
17 CALL LBL 10	Call machining operation
18 L Z+100 R0 FMAX M2	Retract the tool, end program
19 LBL 10	Subprogram 10: Machining operation
20 CYCL DEF 7.0 DATUM SHIFT	Shift datum to center of ellipse
21 CYCL DEF 7.1 X+Q1	
22 CYCL DEF 7.2 Y+Q2	
23 CYCL DEF 10.0 ROTATION	Account for rotational position in the plane
24 CYCL DEF 10.1 ROT+Q8	
25 Q35 = (Q6 -Q5) / Q7	Calculate angle increment
26 Q36 = Q5	Copy starting angle
27 Q37 = 0	Set counter

28 Q21 = Q3 *COS Q36	Calculate X coordinate for starting point
29 Q22 = Q4 *SIN Q36	Calculate Y coordinate for starting point
30 L X+Q21 Y+Q22 R0 FMAX M3	Move to starting point in the plane
31 L Z+Q12 R0 FMAX	Pre-position in spindle axis to set-up clearance
32 L Z-Q9 R0 FQ10	Move to working depth
33 LBL1	
34 Q36 = Q36 +Q35	Update the angle
35 Q37 = Q37 +1	Update the counter
36 Q21 = Q3 *COS Q36	Calculate the current X coordinate
37 Q22 = Q4 *SIN Q36	Calculate the current Y coordinate
38 L X+Q21 Y+Q22 R0 FQ11	Move to next point
39 FN 12: IF +Q37 LT +Q7 GOTO LBL 1	Unfinished? If not finished, return to LBL 1
40 CYCL DEF 10.0 ROTATION	Reset the rotation
41 CYCL DEF 10.1 ROT+0	
42 CYCL DEF 7.0 DATUM SHIFT	Reset the datum shift
43 CYCL DEF 7.1 X+0	
44 CYCL DEF 7.2 Y+0	
45 L Z+Q12 R0 FMAX	Move to set-up clearance
46 LBL 0	End of subprogram
47 END PGM ELLIPSE MM	

Example: Concave cylinder machined with Ball-nose cutter

Program run

- This NC program works only with a Ball-nose cutter. The tool length is measured from the sphere center
- The contour of the cylinder is approximated by many short line segments (defined in Q13). The more line segments you define, the smoother the curve becomes.
- The cylinder is milled in longitudinal cuts (here: parallel to the Y axis).
- The milling direction is determined with the starting angle and end angle in space:
  Machining direction clockwise:
  Starting angle > end angle
  Machining direction counterclockwise:
  Starting angle < end angle
- The tool radius is compensated automatically



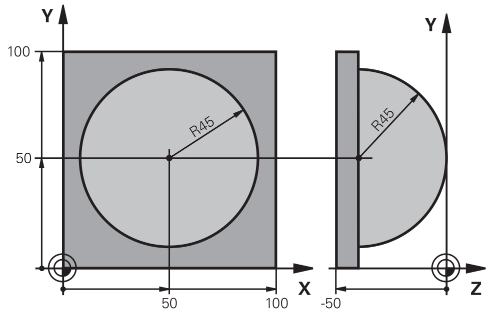
0 BEGIN PGM CYLIN MM	
1 FN 0: Q1 = +50	Center in X axis
2 FN 0: Q2 = +0	Center in Y axis
3 FN 0: Q3 = +0	Center in Z axis
4 FN 0: Q4 = +90	Starting angle in space (Z/X plane)
5 FN 0: Q5 = +270	End angle in space (Z/X plane)
6 FN 0: Q6 = +40	Cylinder radius
7 FN 0: Q7 = +100	Length of the cylinder
8 FN 0: Q8 = +0	Rotational position in the X/Y plane
9 FN 0: Q10 = +5	Allowance for cylinder radius
10 FN 0: Q11 = +250	Feed rate for plunging
11 FN 0: Q12 = +400	Feed rate for milling
12 FN 0: Q13 = +90	Number of cuts
13 BLK FORM 0.1 Z X+0 Y+0 Z-50	Workpiece blank definition
14 BLK FORM 0.2 X+100 Y+100 Z+0	
15 TOOL CALL 1 Z S4000	Tool call
16 L Z+250 R0 FMAX	Retract the tool
17 CALL LBL 10	Call machining operation
18 FN 0: Q10 = +0	Reset allowance
19 CALL LBL 10	Call machining operation
20 L Z+100 R0 FMAX M2	Retract the tool, end program

21 LBL 10	Subprogram 10: Machining operation
22 Q16 = Q6 -Q10 - Q108	Account for allowance and tool, based on the cylinder radius
23 FN 0: Q20 = +1	Set counter
24 FN 0: Q24 = +Q4	Copy starting angle in space (Z/X plane)
25 Q25 = (Q5 -Q4) / Q13	Calculate angle increment
26 CYCL DEF 7.0 DATUM SHIFT	Shift datum to center of cylinder (X axis)
27 CYCL DEF 7.1 X+Q1	
28 CYCL DEF 7.2 Y+Q2	
29 CYCL DEF 7.3 Z+Q3	
30 CYCL DEF 10.0 ROTATION	Account for rotational position in the plane
31 CYCL DEF 10.1 ROT+Q8	
32 L X+0 Y+0 R0 FMAX	Pre-position in the plane to the cylinder center
33 L Z+5 R0 F1000 M3	Pre-position in the spindle axis
34 LBL 1	
35 CC Z+0 X+0	Set pole in the Z/X plane
36 LP PR+Q16 PA+Q24 FQ11	Move to starting position on cylinder, plunge-cutting obliquely into the material
37 L Y+Q7 R0 FQ12	Longitudinal cut in Y+ direction
38 FN 1: Q20 = +Q20 + +1	Update the counter
39 FN 1: Q24 = +Q24 + +Q25	Update solid angle
40 FN 11: IF +Q20 GT +Q13 GOTO LBL 99	Finished? If finished, jump to end
41 LP PR+Q16 PA+Q24 FQ11	Move on an approximated arc for the next longitudinal cut
42 L Y+0 R0 FQ12	Longitudinal cut in Y– direction
43 FN 1: Q20 = +Q20 + +1	Update the counter
44 FN 1: Q24 = +Q24 + +Q25	Update solid angle
45 FN 12: IF +Q20 LT +Q13 GOTO LBL 1	Unfinished? If not finished, return to LBL 1
46 LBL 99	
47 CYCL DEF 10.0 ROTATION	Reset the rotation
48 CYCL DEF 10.1 ROT+0	
49 CYCL DEF 7.0 DATUM SHIFT	Reset the datum shift
50 CYCL DEF 7.1 X+0	
51 CYCL DEF 7.2 Y+0	
52 CYCL DEF 7.3 Z+0	
53 LBL 0	End of subprogram
54 END PGM CYLIN	

Example: Convex sphere machined with end mill

Program run

- NC program requires an end mill.
- The contour of the sphere is approximated by many short lines (in the Z/X plane, defined in Q14). The smaller you define the angle increment, the smoother the curve becomes.
- You can determine the number of contour cuts through the angle increment in the plane (defined in Q18).
- The tool moves upward in three-dimensional cuts.
- The tool radius is compensated automatically



0 BEGIN PGM SPHERE MM	
1 FN 0: Q1 = +50	Center in X axis
2 FN 0: Q2 = +50	Center in Y axis
3 FN 0: Q4 = +90	Starting angle in space (Z/X plane)
4 FN 0: Q5 = +0	End angle in space (Z/X plane)
5 FN 0: Q14 = +5	Angle increment in space
6 FN 0: Q6 = +45	Sphere radius
7 FN 0: Q8 = +0	Starting angle of rotational position in the X/Y plane
8 FN 0: Q9 = +360	End angle of rotational position in the X/Y plane
9 FN 0: Q18 = +10	Angle increment in the X/Y plane for roughing
10 FN 0: Q10 = +5	Allowance in sphere radius for roughing
11 FN 0: Q11 = +2	Set-up clearance for pre-positioning in the spindle axis
12 FN 0: Q12 = +350	Feed rate for milling
13 BLK FORM 0.1 Z X+0 Y+0 Z-50	Workpiece blank definition
14 BLK FORM 0.2 X+100 Y+100 Z+0	
15 TOOL CALL 1 Z S4000	Tool call
16 L Z+250 R0 FMAX	Retract the tool
17 CALL LBL 10	Call machining operation
18 FN 0: Q10 = +0	Reset allowance
19 FN 0: Q18 = +5	Angle increment in the X/Y plane for finishing
20 CALL LBL 10	Call machining operation
21 L Z+100 R0 FMAX M2	Retract the tool, end program
22 LBL 10	Subprogram 10: Machining operation
23 FN 1: Q23 = +q11 + +q6	Calculate Z coordinate for pre-positioning
24 FN 0: Q24 = +Q4	Copy starting angle in space (Z/X plane)
25 FN 1: Q26 = +Q6 + +Q108	Compensate sphere radius for pre-positioning
26 FN 0: Q28 = +Q8	Copy rotational position in the plane
27 FN 1: Q16 = +Q6 + -Q10	Account for allowance in the sphere radius
28 CYCL DEF 7.0 DATUM SHIFT	Shift datum to center of sphere
29 CYCL DEF 7.1 X+Q1	
30 CYCL DEF 7.2 Y+Q2	

31 CYCL DEF 7.3 Z-Q16	
32 CYCL DEF 10.0 ROTATION	Account for starting angle of rotational position in the plane
33 CYCL DEF 10.1 ROT+Q8	
34 LBL 1	Pre-position in the spindle axis
35 CC X+0 Y+0	Set pole in the X/Y plane for pre-positioning
36 LP PR+Q26 PA+Q8 R0 FQ12	Pre-position in the plane
37 CC Z+0 X+Q108	Set pole in the Z/X plane, offset by the tool radius
38 L Y+0 Z+0 FQ12	Move to working depth
39 LBL 2	
40 LP PR+Q6 PA+Q24 FQ12	Move upward on an approximated arc
41 FN 2: Q24 = +Q24 - +Q14	Update solid angle
42 FN 11: IF +Q24 GT +Q5 GOTO LBL 2	Inquire whether an arc is finished. If not finished, return to LBL 2
43 LP PR+Q6 PA+Q5	Move to the end angle in space
44 L Z+Q23 R0 F1000	Retract in the spindle axis
45 L X+Q26 R0 FMAX	Pre-position for next arc
46 FN 1: Q28 = +Q28 + +Q18	Update rotational position in the plane
47 FN 0: Q24 = +Q4	Reset solid angle
48 CYCL DEF 10.0 ROTATION	Activate new rotational position
49 CYCL DEF 10.0 ROT+Q28	
50 FN 12: IF +Q28 LT +Q9 GOTO LBL 1	
51 FN 9: IF +Q28 EQU +Q9 GOTO LBL 1	Unfinished? If not finished, return to LBL 1
52 CYCL DEF 10.0 ROTATION	Reset the rotation
53 CYCL DEF 10.1 ROT+0	
54 CYCL DEF 7.0 DATUM SHIFT	Reset datum shift
55 CYCL DEF 7.1 X+0	
56 CYCL DEF 7.2 Y+0	
57 CYCL DEF 7.3 Z+0	
58 LBL 0	End of subprogram
59 END PGM SPHERE MM	

10

Special Functions

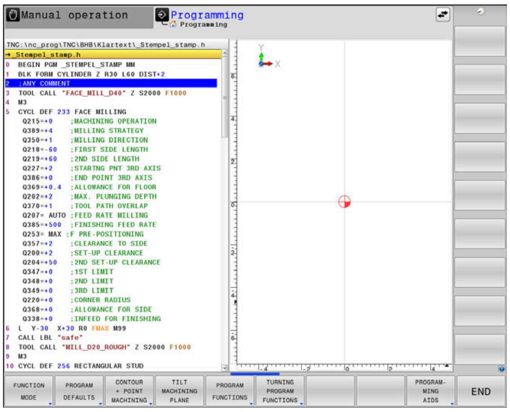
10.1 Overview of special functions

The control provides the following powerful special functions for a large number of applications:

Function	Description
Dynamic Collision Monitoring with integrated fixture management (option 40)	Page 356
Adaptive Feed Control AFC (option 45)	Page 359
Active Chatter Control (option 145)	See the User's Manual for Setup, Testing and Running NC Programs
Working with text files	Page 383
Working with freely definable tables	Page 387

Press the **SPEC FCT** key and the corresponding soft keys to access further special functions of the control. The following tables give you an overview of which functions are available.

Main menu for SPEC FCT special functions



SPEC FCT

- Press the **SPEC FCT** key to select the special functions

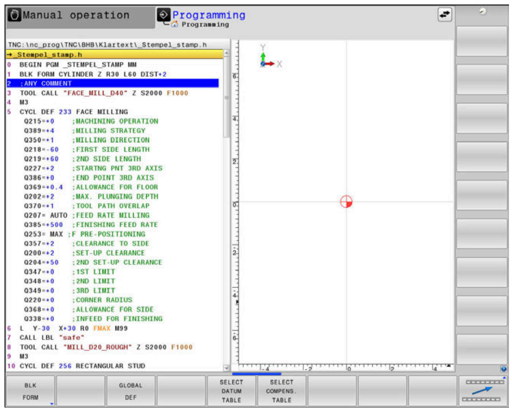
Soft key	Function	Description
FUNCTION MODE	Select machining mode or kinematics	Page 355
PROGRAM DEFAULTS	Define program defaults	Page 353
CONTOUR + POINT MACHINING	Functions for contour and point machining	Page 353
TILT MACHINING PLANE	Define the **PLANE** function	Page 406
PROGRAM FUNCTIONS	Define different conversational functions	Page 354
TURNING PROGRAM FUNCTIONS	Define turning functions	Page 511
PROGRAM-MING AIDS	Programming aids	Page 189

After pressing the **SPEC FCT** key, you can open the **smartSelect** selection window with the **GOTO** key. The control displays a structure overview with all available functions. You can rapidly navigate with the cursor or mouse and select functions in the tree diagram. The control displays online help for the selected function in the window on the right.

Program defaults menu

PROGRAM DEFAULTS

- Press the Program Defaults soft key

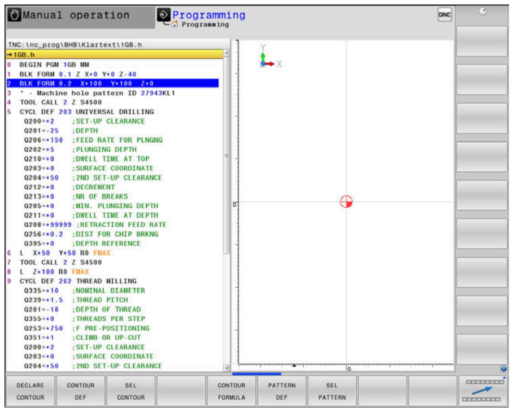


Soft key	Function	Description
BLK FORM	Define workpiece blank	Page 90
DATUM TABLE	Select datum table	See Cycle-Programming User's Manual
SELECT COMPENS. TABLE	Select compensation table	Page 379
GLOBAL DEF	Define global cycle parameters	See Cycle-Programming User's Manual

Functions for contour and point machining menu

CONTOUR + POINT MACHINING

- Press the soft key for functions for contour and point machining

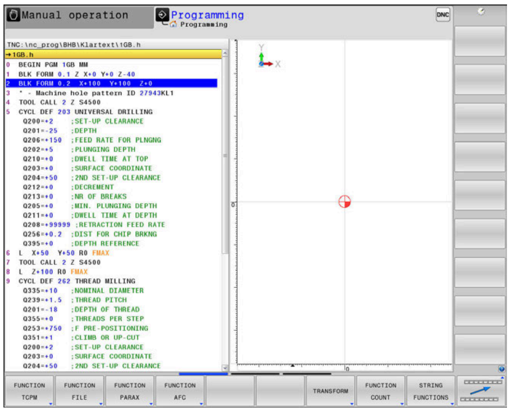


Soft key	Function	Description
DECLARE CONTOUR	Assign contour description	See Cycle-Programming User's Manual
CONTOUR DEF	Define a simple contour formula	See Cycle-Programming User's Manual
SEL CONTOUR	Select a contour definition	See Cycle-Programming User's Manual
CONTOUR FORMULA	Define a complex contour formula	See Cycle-Programming User's Manual
PATTERN DEF	Define regular machining pattern	See Cycle-Programming User's Manual
SEL PATTERN	Select the point file with machining positions	See Cycle-Programming User's Manual

Menu for defining different conversional functions

PROGRAM FUNCTIONS

- Press the **PROGRAM FUNCTIONS** soft key



Soft key	Function	Description
FUNCTION TCPM	Define the positioning behavior for rotary axes	Page 443
FUNCTION FILE	Define file functions	Page 372
FUNCTION PARAX	Define the positioning behavior for parallel axes U, V, W	Page 364
FUNCTION AFC	Define Adaptive Feed Control	Page 359
TRANSFORM / CORRDATA	Define coordinate transformations	Page 373
FUNCTION COUNT	Define the counter	Page 381
STRING FUNCTIONS	Define string functions	Page 324
FUNCTION DRESS	Define dressing mode	Page 542
FUNCTION SPINDLE	Define pulsing spindle speed	Page 392
FUNCTION FEED	Define recurring dwell time	Page 394
FUNCTION DCM	Define Dynamic Collision Monitoring DCM	Page 356
FUNCTION DWELL	Define dwell time in seconds or revolutions	Page 396
FUNCTION LIFTOFF	Lift off tool at NC stop	Page 397
INSERT COMMENT	Add comments	Page 192
FUNCTION PROG PATH	Choose path interpretation	Page 457

10.2 Function mode

Program function mode

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

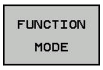
To switch between milling and turning operations, you must switch to the respective mode.

If your machine manufacturer has enabled the selection of various kinematic models, then you can switch between them using the **FUNCTION MODE** soft key.

Procedure

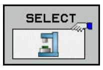
To switch the kinematic model, proceed as follows:

- Show the soft-key row for special functions



- Press the **FUNCTION MODE** soft key

- Press the **MILL** soft key

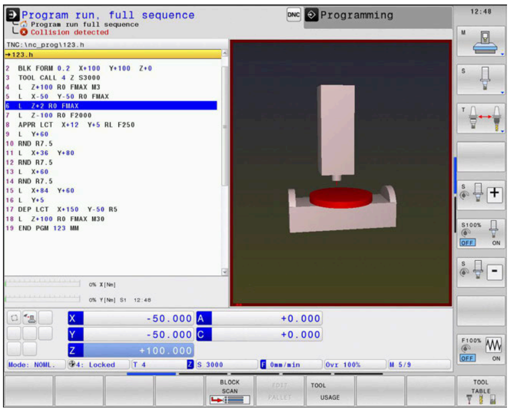


- Press the **SELECT KINEMATICS** soft key
- Select the desired kinematic model

10.3 Dynamic Collision Monitoring (option 40)

Function

> Refer to your machine manual!
> The machine tool builder needs to adapt the **Dynamic Collision Monitoring (DCM)** function to the control.



The machine manufacturer can define any objects that will be monitored by the control during all machining operations. If two objects monitored for collision come within a defined distance of each other, the control generates an error message and terminates the movement.

The control also monitors the active tool for collision and displays the situation graphically. The control always assumes cylindrical tools. The control likewise monitors stepped tools according to their definition in the tool table.

The control takes into account the following definitions from the tool table:

- Tool lengths
- Tool radii
- Tool dimensions
- Tool carrier kinematics

NOTICE

Danger of collision!

Even if **Dynamic Collision Monitoring (DCM)** is active, the control does not automatically monitor the workpiece for collisions, be it with the tool or with other machine components. There is a danger of collision during machining!

- Check the machining sequence using a graphic simulation
- Carefully test the NC program or program section in the **Program run, single block** operating mode

Generally valid constraints:

- The **Dynamic Collision Monitoring (DCM)** function helps to reduce the danger of collision. However, the control cannot consider all possible constellations during operation.
- The control can only protect those machine components from collision that your machine tool builder has defined correctly with regard to dimensions, orientation and position.
- The control can only monitor tools for which you have defined **positive tool radii** and **positive tool lengths** in the tool table.
- When a touch probe cycle starts, the control no longer monitors the stylus length and ball-tip diameter so that you can also probe collision objects.
- For certain tools (such as face milling cutters), the radius that would cause a collision can be greater than the value defined in the tool table.
- **DL** and **DR** tool oversizes from the tool table are taken into account by the control. Tool oversizes from the **TOOL CALL** block are not accounted for.

Activating and deactivating collision monitoring in the NC program

In some cases it is necessary to temporarily deactivate collision monitoring:

- To reduce the distance between two objects monitored for collision
- To prevent stops during program runs

NOTICE

Danger of collision!

If the **Dynamic Collision Monitoring (DCM)** function is inactive, the control does not perform any automatic collision checking. This means that movements that might cause collisions will not be prevented. There is a danger of collision during all movements!

- ▶ Make sure to activate collision monitoring whenever possible
- ▶ Make sure to always re-activate collision monitoring after a temporary deactivation
- ▶ With collision monitoring deactivated, carefully test the NC program or program section in the **Program run, single block** operating mode

Temporarily activating and deactivating collision monitoring via program control

- Open the NC program in **Programming** mode
- Place the cursor at the desired position, e.g. before Cycle 800 to enable eccentric turning

SPEC FCT

- Press the **SPEC FCT** key

PROGRAM FUNCTIONS

- Press the **PROGRAM FUNCTIONS** soft key

▷

- Shift the soft-key row

FUNCTION DCM

- Press the **FUNCTION DCM** soft key

FUNCTION DCM OFF

FUNCTION DCM ON

- Select the condition with the corresponding soft key:
  - **FUNCTION DCM OFF**: This NC command temporarily deactivates collision monitoring. The deactivation is effective only until the end of the main program or until the next **FUNCTION DCM ON**. When another NC program is called, DCM is active again.
  - **FUNCTION DCM ON**: This NC command cancels an existing **FUNCTION DCM OFF**.

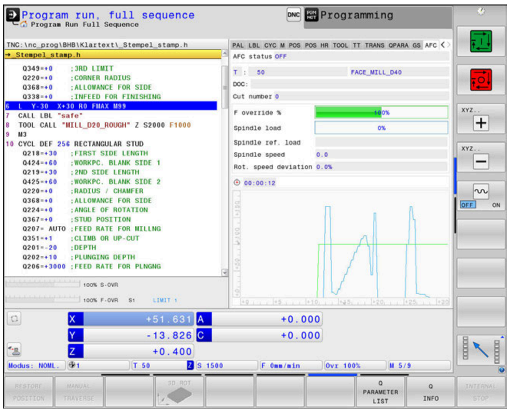
The settings applied with the **FUNCTION DCM** function are only effective in the active NC program.

After terminating the program run or selecting a new NC program, the settings made for **Program run** and **Manual operation** with the **COLLISION** soft key become effective again.

Further information: User's Manual for Setup, Testing and Running NC Programs

10.4 Adaptive Feed Control (AFC) (option 45)

Application



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

Your machine tool builder may also specify whether the spindle power or any other value is used as input quantity by the control.

If you have enabled the software option for turning (Option 50), you can use AFC in turning mode as well.

Adaptive feed control is not intended for tools with diameters less than 5 mm. If the rated power consumption of the spindle is very high, the limit diameter of the tool may be larger.

Do not work with adaptive feed control in operations in which the feed rate and spindle speed must be adapted to each other, such as tapping.

In adaptive feed control the control automatically controls the feed rate during NC program run as a function of the current spindle power. The spindle power required for each machining step is to be determined in a teach-in cut and saved by the control in a file belonging to the NC program. At the start of each machining step, usually when the spindle is switched on, the control controls the feed rate so that it remains within the limits that you have defined.

If the cutting conditions do not change, you can define the spindle power consumption, which has been determined in a teach-in cut, as permanent tool-dependent reference power. Use the **AFC-LOAD** column in the tool table to do this. If you enter a value manually in this column, the control does not execute any more teach-in cuts.

This makes it possible to avoid negative effects on the tool, the workpiece, and the machine that might be caused by changing cutting conditions. Cutting conditions are changed particularly by:

- Tool wear
- Fluctuating cutting depths that occur especially with cast parts
- Fluctuating hardness caused by material flaws

Adaptive feed control (AFC) has the following advantages:

- Optimization of machining time

  By controlling the feed rate, the control tries to maintain the previously recorded maximum spindle power or the reference power specified in the tool table (**AFC-LOAD** column) during the entire machining time. It shortens the machining time by increasing the feed rate in machining zones with little material removal.
- Tool monitoring

  If the spindle power exceeds the recorded or specified maximum value (**AFC-LOAD** column of the tool table), the control decreases the feed rate until the reference spindle power is reached again. If the maximum spindle power is exceeded during machining and at the same time the feed rate falls below the minimum that you have defined, the control reacts by shutting down. This helps to prevent further damage after a tool breaks or is worn out.
- Protection of the machine's mechanical elements

  Timely feed rate reduction and shutdown responses help to avoid machine overload.

Defining basic AFC settings

In the **AFC.TAB** table, which must be saved in the **TNC:\table** directory, you enter the control settings with which the control performs the feed rate control.

The data in this table are default values that are copied into a file belonging to the respective NC program during a teach-in cut. The values act as the basis for feedback control.

> If you define a tool-specific feedback-control reference power using the **AFC-LOAD** column in the tool table, the control generates the associated file for the relevant NC program without a teach-in cut. The file is created shortly before feedback control becomes effective.

Enter the following data in the table:

Column	Function
NR	Consecutive line number in the table (has no further functions)
AFC	Name of the control setting. You enter this name in the **AFC** column of the tool table. It specifies the assignment of control parameters to the tool.
FMIN	Feed rate at which the control is to conduct a shutdown response. Enter the value in percent with respect to the programmed feed rate. Input range: 50 to 100 %
FMAX	Maximum feed rate in the material up to which the control can automatically increase the feed rate. Enter the value in percent of the programmed feed rate.
FIDL	Feed rate for traverse when the tool is not cutting. Enter the value in percent of the programmed feed rate.
FENT	Feed rate for traverse when the tool moves into or out of the material. Enter the value in percent with respect to the programmed feed rate. Maximum input value: 100 %
OVLD	Reaction that the control is to perform in case of overload: ■ **M**: Execution of a macro defined by the machine tool builder ■ **S**: Immediate NC stop ■ **F**: NC stop if the tool has been retracted ■ **E**: Just display an error message on the screen ■ **L**: Disable active tool ■ **-**: No overload reaction The control performs the selected overload reaction if, when feedback control is active, the maximum spindle power is exceeded for more than one second and at the same time the feed rate falls below the minimum you defined. Enter the desired function via the alphabetic keyboard. In conjunction with the cut-related tool wear monitoring function, the control will only evaluate the options **M**, **E**, and **L**! **Further information:** User's Manual for Setup, Testing and Running NC Programs
POUT	Spindle power at which the control is to detect that the tool moves out of the workpiece. Enter the value in percent of the learned reference load. Recommended input value: 8 %
SENS	Sensitivity (aggressiveness) of feedback control. A value between 50 and 200 can be entered. 50 is for slow control, 200 for a very aggressive control. An aggressive control reacts quickly and with strong changes to the values, but it tends to overshoot. Recommended value: 100
PLC	Value that the control is to transfer to the PLC at the beginning of a machining step. The machine manufacturer defines the function, so refer to your machine manual.

In the **AFC.TAB** table you can define as many control settings (lines) as desired.

If there is no AFC.TAB table in the **TNC:\table** directory, the control uses a fixed control setting for the teach-in cut. If, alternatively, a tool-dependent reference power value exists, the control uses it immediately. HEIDENHAIN recommends to use the AFC.TAB table in order to ensure a safe and well-defined operation.

Proceed as follows to create the AFC.TAB file (only necessary if the file does not yet exist):

- Select the **Programming** operating mode
- To call the file manager, press the **PGM MGT** key
- Select the **TNC:** directory
- Create a new **AFC.TAB** file
- Press the **ENT** key
- > The control displays a list with table formats.
- Select the **AFC.TAB** table format and confirm with the **ENT** key
- > The control creates the table that contains the control settings.

Programming AFC

NOTICE

Caution: Danger to the tool and workpiece!

If you activate the **FUNCTION MODE TURN** machining mode, the control will clear the current **OVLD** values. This means that you need to program the machining mode before the tool call! If the programming sequence is not correct, no tool monitoring will take place, which might result in damage to the tool or workpiece!

- Program the **FUNCTION MODE TURN** machining mode before the tool call

Proceed as follows to program the AFC functions for starting and ending the teach in cut:

SPEC FCT
- Press the **SPEC FCT** key

PROGRAM FUNCTIONS
- Press the **PROGRAM FUNCTIONS** soft key

FUNCTION AFC
- Press the **FUNCTION AFC** soft key
- Select the function

The control provides several functions that enable you to start and stop AFC:

- **FUNCTION AFC CTRL**: The **AFC CTRL** function activates feedback control mode starting with this NC block, even if the learning phase has not been completed yet.
- **FUNCTION AFC CUT BEGIN TIME1 DIST2 LOAD3**: The control starts a sequence of cuts with active **AFC**. The changeover from the teach-in cut to feedback control mode begins as soon as the reference power has been determined in the teach-in phase, or once one of the **TIME**, **DIST** or **LOAD** conditions has been met.
  - With **TIME**, you define the maximum duration of the teach-in phase in seconds.
  - **DIST** defines the maximum distance for the teach-in cut.
  - With **LOAD**, you can set a reference load directly. If you enter a reference load > 100 %, the control automatically limits the value to 100 %.
- **FUNCTION AFC CUT END**: The **AFC CUT END** function deactivates the AFC control.

The **TIME**, **DIST** and **LOAD** defaults are modally effective. They can be reset by entering **0**.

You can define a feedback-control reference power with the **AFC LOAD** tool table column and the **LOAD** input in the NC program. You can activate the **AFC LOAD** value via the tool call and the **LOAD** value with the **FUNCTION AFC CUT BEGIN** function.

If you program both values, the control will use the value programmed in the NC program!

Opening the AFC table

With a teach-in cut, the control at first copies the basic settings for each machining step, as defined in the AFC.TAB table, to a file called **<name>.H.AFC.DEP**. **<name>** is the name of the NC program for which you have recorded the teach-in cut. In addition, the control measures the maximum spindle power consumed during the teach-in cut and saves this value in the table.

You can change the **<name>.H.AFC.DEP** file in **Programming** operating mode.

If necessary, you can even delete a machining step (entire line) there.

The **dependentFiles** machine parameter (no. 122101) must be set to **MANUAL** so that you can view the dependent files in the file manager.

In order to edit the **<name>.H.AFC.DEP** file, you must first set the file manager so that all file types can be displayed (**SELECT TYPE** soft key).

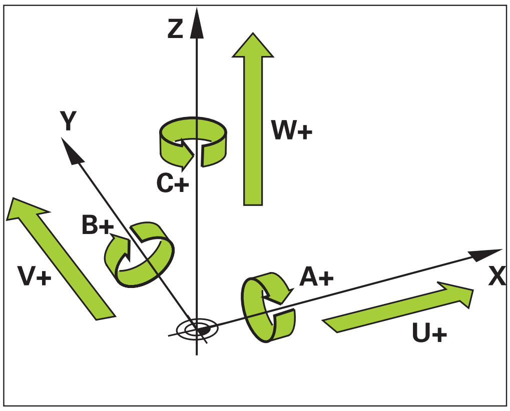
Further information: "Files", Page 103

Further information: User's Manual for Setup, Testing and Running NC Programs

10.5 Working with the parallel axes U, V and W

Overview

> Refer to your machine manual!
> Your machine must be configured by the machine manufacturer if you want to use parallel-axis functions.
> The number, designation and assignment of the programmable axes depend on the machine.



In addition to the principal axes X, Y and Z, the parallel axes U, V and W are available.

The principal axes and parallel axes are usually assigned to each other as follows:

Principal axis	Parallel axis	Rotary axis
X	U	A
Y	V	B
Z	W	C

The control provides the following functions for machining with the parallel axes U, V and W:

Soft key	Function	Meaning	Page
FUNCTION PARAXCOMP	**PARAXCOMP**	Define the control's behavior when positioning parallel axes	367
FUNCTION PARAXMODE	**PARAXMODE**	Define the axes the control is to use for machining	368

> You must deactivate the parallel-axis functions before switching the machine kinematics.
> You can deactivate the programming of parallel axes with the machine parameter **noParaxMode** (no. 105413).

Automatic calculation of the parallel axes

In machine parameter **parAxComp** (no. 300205), your machine tool builder specifies whether the parallel axis function is active by default.

After the control has been started up, the configuration defined by the machine tool builder is effective.

If the machine tool builder has already enabled the parallel axis in the configuration, the control takes this axis into account in the calculations, without you having to program **PARAXCOMP**.

This means that the control permanently takes the parallel axis into account in the calculations and you can therefore also probe a workpiece with any position of the W axis, for example.

Please note that **PARAXCOMP OFF** does not deactivate the parallel axis in this case, but the control reactivates the standard configuration.

The control deactivates automatic calculation only if you include the axis in the NC block, e.g. **PARAXCOMP OFF W**.

FUNCTION PARAXCOMP DISPLAY

Example

```
13 FUNCTION PARAXCOMP  DISPLAY W
```

Use the **PARAXCOMP DISPLAY** function to activate the display function for parallel axis movements. The control includes movements of the parallel axis in the position display of the associated principal axis (sum display). Therefore, the position display of the principal axis always displays the relative distance from the tool to the workpiece, regardless of whether you move the principal axis or the minor axis.

Proceed as follows for the definition:

SPEC FCT
- Show the soft-key row with special functions

PROGRAM FUNCTIONS
- Press the **PROGRAM FUNCTIONS** soft key

FUNCTION PARAX
- Press the **FUNCTION PARAX** soft key

FUNCTION PARAXCOMP
- Press the **FUNCTION PARAXCOMP** soft key

FUNCTION PARAXCOMP DISPLAY
- Select the **FUNCTION PARAXCOMP DISPLAY** function
- Define the parallel axis whose movements the control is to take into account in the position display of the associated principal axis

FUNCTION PARAXCOMP MOVE

Example

```
13 FUNCTION PARAXCOMP  MOVE W
```

> The **PARAXCOMP MOVE** function can be used only in connection with straight-line blocks (**L**).

The control uses the **PARAXCOMP MOVE** function to compensate for movements of a parallel axis by performing compensation movements in the associated principal axis.

For example, if a parallel-axis movement is performed in the negative W-axis direction, the principal axis Z is moved simultaneously in the positive direction by the same value. The relative distance from the tool to the workpiece remains the same. Application in gantry-type milling machines: Retract the spindle sleeve to move the cross beam down simultaneously.

Proceed as follows for the definition:

SPEC FCT
- Show the soft-key row with special functions

PROGRAM FUNCTIONS
- Press the **PROGRAM FUNCTIONS** soft key

FUNCTION PARAX
- Press the **FUNCTION PARAX** soft key

FUNCTION PARAXCOMP
- Press the **FUNCTION PARAXCOMP** soft key

FUNCTION PARAXCOMP MOVE
- Select the **FUNCTION PARAXCOMP MOVE** function
- Define the parallel axis

> Possible offset values (U_OFFS, V_OFFS and W_OFFS from the preset table) to be taken into account will be specified by your machine tool builder in the **presetToAlignAxis** machine parameter (no. 300203).

Deactivating FUNCTION PARAXCOMP

> After the control has been started up, the configuration defined by the machine tool builder is effective.
>
> The **PARAXCOMP** parallel-axis function is automatically reset by the control with the following functions:
>
> - Selection of NC program
> - **PARAXCOMP OFF**
>
> You must deactivate the parallel-axis functions before switching the machine kinematics.

Example

```
13 FUNCTION PARAXCOMP  OFF
13 FUNCTION PARAXCOMP  OFF W
```

Use the **PARAXCOMP OFF** function to switch off the **PARAXCOMP DISPLAY** and **PARAXCOMP MOVE** parallel-axis functions. Proceed as follows for the definition:

SPEC FCT
- Show the soft-key row with special functions

PROGRAM FUNCTIONS
- Press the **PROGRAM FUNCTIONS** soft key

FUNCTION PARAX
- Press the **FUNCTION PARAX** soft key

FUNCTION PARAXCOMP
- Press the **FUNCTION PARAXCOMP** soft key

FUNCTION PARAXCOMP OFF
- Select **FUNCTION PARAXCOMP OFF**
- Enter an axis, if required

> Your machine tool builder can activate the **PARAXCOMP** function permanently with a machine parameter.
>
> If you want to switch the function off, you must indicate the parallel axis in the NC block, for example **FUNCTION PARAXCOMP OFF W**.
>
> **Further information:** "Automatic calculation of the parallel axes", Page 365

FUNCTION PARAXMODE

Example

13 FUNCTION PARAXMODE X Y W

To activate the **PARAXMODE** function, you must always define three axes.

If your machine tool builder has not yet activated the **PARAXCOMP** function as default, you must activate **PARAXCOMP** before you can work with **PARAXMODE**.

In order for the control to offset the principal axis deselected with **PARAXMODE**, switch the **PARAXCOMP** function on for this axis.

Use the **PARAXMODE** function to define the axes the control is to use for machining. You program all traverse movements and contour descriptions in the principal axes X, Y and Z, independent of your machine.

Define 3 axes in the **PARAXMODE** function (e.g. **FUNCTION PARAXMODE X Y W**) to be used by the control for programmed traverse movements.

Proceed as follows for the definition:

SPEC FCT
- Show the soft-key row with special functions

PROGRAM FUNCTIONS
- Press the **PROGRAM FUNCTIONS** soft key

FUNCTION PARAX
- Press the **FUNCTION PARAX** soft key

FUNCTION PARAXMODE
- Press the **FUNCTION PARAXMODE** soft key

FUNCTION PARAXMODE
- Select **FUNCTION PARAXMODE**
- Define the axes for machining

Moving the principal axis and the parallel axis

Example

```
13 FUNCTION PARAXMODE  X Y W
14 L  Z+100 &Z+150 R0 FMAX
```

If the **PARAXMODE** function is active, the control uses the axes defined in the function to execute the programmed traverse movements. If the control is to move the principal axis deselected by **PARAXMODE**, you can identify this axis by additionally entering the character **&**. The **&** character then refers to the principal axis.

Proceed as follows:

- Press the **L** key
- > The control opens a linear block.
- Define coordinates
- Define radius compensation
- Press the left arrow key
- > The control shows the **&Z** character.
- If applicable, use the axis-direction keys to select the axis
- Define coordinate
- Press the **ENT** key

> The **&** syntax element is only permitted in L blocks.
>
> Additional positioning of a principal axis with the **&** command is done in the REF system. If you have set the position display to display ACTUAL values, this movement will not be shown. If necessary, switch the position display to REF values.
>
> Your machine tool builder will define the calculation of possible offset values (X_OFFS, Y_OFFS and Z_OFFS from the preset table) for the axes positioned with the **&** operator in the **presetToAlignAxis** machine parameter (no. 300203).

Deactivating FUNCTION PARAXMODE

After the control has been started up, the configuration defined by the machine tool builder is effective.

The control automatically resets the **PARAXMODE OFF** parallel-axis function via the following functions:

- Selection of NC program
- End of program
- **M2** and **M30**
- **PARAXMODE OFF**

You must deactivate the parallel-axis functions before switching the machine kinematics.

Example

```
13 FUNCTION PARAXMODE  OFF
```

Use the **PARAXCOMP OFF** function to switch off the parallel-axis function. The control then uses the principal axes defined by the machine manufacturer. Proceed as follows for the definition:

SPEC FCT
- Show the soft-key row with special functions

PROGRAM FUNCTIONS
- Press the **PROGRAM FUNCTIONS** soft key

FUNCTION PARAX
- Press the **FUNCTION PARAX** soft key

FUNCTION PARAXMODE
- Press the **FUNCTION PARAXMODE** soft key

FUNCTION PARAXMODE OFF
- Select **FUNCTION PARAXMODE OFF**

Example: Drilling with the W axis

0 BEGIN PGM PAR MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 5 Z S2222	Call the tool in the spindle axis Z
4 L Z+100 R0 FMAX M3	Position the principal axis
5 CYCL DEF 200 DRILLING	
Q200=+2 ;SET-UP CLEARANCE	
Q201=-20 ;DEPTH	
Q206=+150 ;FEED RATE FOR PLNGNG	
Q202=+5 ;PLUNGING DEPTH	
Q210=+0 ;DWELL TIME AT TOP	
Q203=+0 ;SURFACE COORDINATE	
Q204=+50 ;2ND SET-UP CLEARANCE	
Q211=+0 ;DWELL TIME AT DEPTH	
Q395=+0 ;DEPTH REFERENCE	
6 FUNCTION PARAXCOMP DISPLAY Z	Activate display compensation
7 FUNCTION PARAXMODE X Y W	Positive axis selection
8 L X+50 Y+50 R0 FMAX M99	Infeed runs minor axis W
9 FUNCTION PARAXMODE OFF	Restore the standard configuration
10 L M30	
11 END PGM PAR MM	

10.6 File functions

Application

With the **FILE FUNCTION** functions, you can copy, move, and delete the file operations from the NC program.

> Programming and operating information:
> - You must not use **FILE** functions on NC programs or files, to which you have previously made reference with functions such as **CALL PGM** or **CYCL DEF 12 PGM CALL**.
> - The **FUNCTION FILE** function is considered only in the **Program run, single block** and **Program run, full sequence** operating modes.

Defining file functions

SPEC FCT
- ▶ Press the special functions key

PROGRAM FUNCTIONS
- ▶ Select the program functions

FUNCTION FILE
- ▶ Select file operations
- > The control displays the available functions.

Soft key	Function	Meaning
FILE COPY	**FILE COPY**	Copy file: Enter the name and path of the file to be copied, as well as the target path
FILE MOVE	**FILE MOVE**	Move file: Enter the name and path of the file to be moved, as well as the target path
FILE DELETE	**FILE DELETE**	Delete file: Enter the path and name of the file to be deleted

If you try to copy a file that does not exist, the control generates an error message.

FILE DELETE does not generate an error message if you try to delete a non-existing file.

10.7 Defining coordinate transformations

Overview

The control offers the following functions for programming coordinate transformations:

Soft key	Meaning
TRANS DATUM	Datum shift
FUNCTION CORRDATA	Select compensation tables
FUNCTION CORRDATA RESET	Reset compensation

TRANS DATUM

As an alternative to the coordinate transformation Cycle 7, **DATUM SHIFT**, you can also use the **TRANS DATUM** Klartext function. Just as in Cycle 7, you can use **TRANS DATUM** to directly program shift values or activate a line from a selectable datum table. In addition, there is also the **TRANS DATUM RESET** function, with which you can easily reset an active datum shift.

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

TRANS DATUM AXIS

Example

```
13 TRANS DATUM AXIS X+10 Y+25 Z+42
```

You can define a datum shift by entering values in the respective axis with the **TRANS DATUM AXIS** function. You can define up to nine coordinates in one NC block, and incremental entries are possible. Proceed as follows for the definition:

SPEC FCT
- Show the soft-key row with special functions

PROGRAM FUNCTIONS
- Press the **PROGRAM FUNCTIONS** soft key

TRANSFORM / CORRDATA
- Press the **TRANSFORM / CORRDATA** soft key

TRANS DATUM
- Press the **TRANS DATUM** soft key

VALUES X Y Z
- Select the value input soft key
- Enter the datum shift in the affected axes, confirming with the **ENT** key each time

> Values entered as absolute numbers refer to the workpiece preset, which is specified either by presetting or by selecting a preset from the preset table.
>
> Incremental values always refer to the datum which was last valid (this may be a datum which has already been shifted).

TRANS DATUM TABLE

Example

```
13 TRANS DATUM TABLE TABLINE25
```

You can define a datum shift by selecting a datum number from a datum table with the **TRANS DATUM TABLE** function. Proceed as follows for the definition:

SPEC FCT
- Show the soft-key row with special functions

PROGRAM FUNCTIONS
- Press the **PROGRAM FUNCTIONS** soft key

TRANSFORM / CORRDATA
- Select transformations

TRANS DATUM
- Select the **TRANS DATUM** datum shift

TABLE .D
- Select the **TRANS DATUM TABLE** datum shift
- Enter the line number to be activated by the control, confirm with the **ENT** key
- If desired, enter the name of the datum table from which you want to activate the datum number, and confirm with the **ENT** key. If you do not want to define a datum table, confirm with the **NO ENT** key

> If you have not defined a datum table in the **TRANS DATUM TABLE** block, then the control uses the datum table previously selected with **SEL TABLE** or the datum table activated in the **Program run, single block** or **Program run, full sequence** operating mode (status **M**).

TRANS DATUM RESET

Example

```
13 TRANS DATUM  RESET
```

Use the **TRANS DATUM RESET** function to cancel a datum shift. How you previously defined the datum is irrelevant. Proceed as follows for the definition:

SPEC FCT	▶ Show the soft-key row with special functions
PROGRAM FUNCTIONS	▶ Press the **PROGRAM FUNCTIONS** soft key
TRANSFORM / CORRDATA	▶ Select transformations
TRANS DATUM	▶ Select the **TRANS DATUM** datum shift
RESET DATUM SHIFT	▶ Press the **RESET DATUM SHIFT** soft key

10.8 Compensation table

Application

With the compensation table, you can save compensations in the tool coordinate system (T-CS) or in the working plane coordinate system (WPL-CS).

The compensation table **.tco** is the alternative to compensating with **DL**, **DR**, and **DR2** in the Tool Call block. As soon you have activated a compensation table, the control overwrites the compensation value from the Tool Call block.

During turning operations, the compensation table ***.tco** is an alternative to programming with **FUNCTION TURNDATA CORR-TCS**; the compensation table ***.wco** is an alternative to **FUNCTION TURNDATA CORR-WPL**.

The compensation tables offer the following benefits:

- Values can be changed without adapting the NC program
- Values can be changed during NC program run

If you change a value, then this change does not become active until the compensation is called again.

Types of compensation tables

Via the file name extension, you can determine in which coordinate system the control will perform the compensation.

The control offers the following compensation options via tables:

- tco (tool correction): Compensation in the tool coordinate system (T-CS)
- wco (workpiece correction): Compensation in the working plane coordinate system (WPL-CS)

Compensation via the table is an alternative to compensation in the TOOL CALL block. Compensation from the table overwrites an already programmed compensation in the TOOL CALL block.

Tool compensation via the ".tco" table

The compensations in the tables with the file name extension ".tco" compensate for the active tool. The table applies to all tool types, which is why, during creation, you also see columns that you may not need for your tool type.

> Enter only those values that are relevant to your tool. If you compensate for values that are not present with the existing tool, the control issues an error message.

The compensations have the following effects:

- In the case of milling cutters, as an alternative to the delta values in the **TOOL CALL**
- In the case of turning tools, as an alternative to **FUNCTION TURNDATA CORR-TCS**
- In the case of grinding tools, as compensation for **LO** and **R-OVR**

Tool compensation via the ".wco" table

The compensations in the tables with the file name extension ".wco" act as a shift in the working plane coordinate system (WPL-CS).

The compensations have the following effects:

- In the case of turning operations, as an alternative to **FUNCTION TURNDATA CORR-WPL**
- An X shift affects the radius

Creating a compensation table

Before you can work with a compensation table, you must first create the respective table.

You can create a compensation table as follows:

- Switch to the **Programming** operating mode

PGM MGT
- Press the **PGM MGT** key

NEW FILE
- Press the **NEW FILE** soft key
- Enter a file name with the desired extension (e.g., Corr.tco)

ENT
- Confirm by pressing the **ENT** key
- Select the unit of measure

ENT
- Confirm by pressing the **ENT** key

APPEND N LINES AT END
- Press the **APPEND N LINES AT END** soft key
- Enter the compensation values

Activate the compensation table

Select compensation table

If you are using compensation tables, then use the function **SEL CORR-TABLE** to activate the desired compensation table from within the NC program.

To add a compensation table to the NC program, proceed as follows:

SPEC FCT
- Press the **SPEC FCT** key

PROGRAM DEFAULTS
- Press the **PROGRAM DEFAULTS** soft key

SELECT COMPENS. TABLE
- Press the **SELECT COMPENS.** Press the **SELECT COMPENS. TABLE** soft key

TCS
- Press the soft key of the table type (e.g., **TCS**)
- Select the table

If you are working without the **SEL CORR-TABLE** function, then you must activate the desired table prior to the test run or program run.

In all operating modes, proceed as follows:

- Select the desired operating mode
- Select the desired table in the file manager
- > In **Test Run** mode, the table receives the status S; in the operating modes **Program run, single block** and **Program run, full sequence**, the table receives the status M.

Activate the compensation value

To activate a compensation value in the NC program, proceed as follows:

SPEC FCT
- Press the **SPEC FCT** key

PROGRAM FUNCTIONS
- Press the **PROGRAM FUNCTIONS** soft key

TRANSFORM / CORRDATA
- Press the **TRANSFORM / CORRDATA** soft key

TRANSFORM / CORRDATA
- Press the **FUNCTION CORRDATA** soft key

TCS
- Press the soft key of the desired compensation (e.g., **TCS**)
- Enter the line number

Duration of effect of the compensation

The activated compensation stays in effect until the end of the program or until a tool change occurs.

With **FUNCTION CORRDATA RESET**, you can program the compensations to reset.

Editing a compensation table during program run

You can change the values in the active compensation table during program run. As long as the compensation table is not yet active, the control dims the soft key.

Proceed as follows:

- Press the **SELECT COMPENS. TABLES** soft key

- Press the soft key of the desired table (e.g., **COMPENS.**)**COMPENS. TABLE T-CS**

- Set the **EDIT** soft key to **ON**
- Use the arrow keys to navigate to the desired location
- Edit the value

> The changed data do not take effect until after the compensation has been activated again.

10.9 Defining a counter

Application

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

The **FUNCTION COUNT** function allows you to control a simple counter from within the NC program. For example, this function allows you to count the number of manufactured workpieces.

Proceed as follows for the definition:

SPEC FCT

- Show the soft key row with special functions

PROGRAM FUNCTIONS

- Press the **PROGRAM FUNCTIONS** soft key

FUNCTION COUNT

- Press the **FUNCTION COUNT** soft key

NOTICE

Caution: Data may be lost!

Only one counter can be managed by the control. If you execute an NC program that resets the counter, any counter progress of another NC program will be deleted.

- Please check prior to machining whether a counter is active.
- If necessary, note down the counter value and enter it again via the MOD menu after execution.

You can use Cycle 225 to engrave the current counter value into the workpiece.
Further information: Cycle Programming User's Manual

Effect in the Test Run operating mode

You can simulate the counter in the **Test Run** operating mode. Only the count you have defined directly in the NC program is effective. The count in the MOD menu remains unaffected.

Effect in the Program Run Single Block and Program Run Full Sequence operating modes

The count from the MOD menu is only effective in the **Program Run Single Block** and **Program Run Full Sequence** operating modes.
The count is retained even after a restart of the control.

Defining FUNCTION COUNT

The **FUNCTION COUNT** function provides the following possibilities:

Soft key	Meaning
FUNCTION COUNT INC	Increase count by 1
FUNCTION COUNT RESET	Reset counter
FUNCTION COUNT TARGET	Set the nominal count (target value) to the desired value Input value: 0–9999
FUNCTION COUNT SET	Set the counter to the desired value Input value: 0–9999
FUNCTION COUNT ADD	Increment the counter by the desired value Input value: 0–9999
FUNCTION COUNT REPEAT	Repeat the NC program starting from this label if more parts are to be machined.

Example

5 FUNCTION COUNT RESET	Reset the counter value
6 FUNCTION COUNT TARGET10	Enter the target number of parts to be machined
7 LBL 11	Enter the jump label
8 L ...	Machining
51 FUNCTION COUNT INC	Increment the counter value
52 FUNCTION COUNT REPEAT LBL 11	Repeat the machining operations if more parts are to be machined.
53 M30	
54 END PGM	

10.10 Creating text files

Application

You can use the control's text editor to write and edit texts. Typical applications:

- Recording test results
- Documenting working procedures
- Creating formula collections

Text files have the extension .A (for ASCII files). If you want to edit other types of files, you must first convert them into type .A files.

Opening and exiting a text file

- Operating mode: Press the **Programming** key
- To call the file manager, press the **PGM MGT** key.
- Display type .A files: Press the **SELECT TYPE** soft key and **SHOW ALL** soft key in succession
- Select a file and open it with the **SELECT** soft key or **ENT** key, or open a new file by entering the new file name and confirming your entry with the **ENT** key

To leave the text editor, call the file manager and select a file of a different file type, for example an NC program.

Soft key	Cursor movements
MOVE WORD →	Move cursor one word to the right
MOVE WORD ←	Move cursor one word to the left
PAGE ↓	Go to next screen page
PAGE ↑	Go to previous screen page
BEGIN ↑	Cursor at beginning of file
END ↓	Cursor at end of file

Editing texts

Above the first line of the text editor, there is an information field showing the file name, location and line information:

File:	Name of the text file
Line:	Line in which the cursor is presently located
Column:	Column in which the cursor is presently located

The text is inserted or overwritten at the location of the cursor. You can move the cursor to any desired position in the text file by pressing the arrow keys.

You can insert a line break with the **RETURN** or **ENT** key.

Deleting and re-inserting characters, words and lines

With the text editor, you can erase words and even lines, and insert them at any desired location in the text.

- Move the cursor to the word or line that you wish to erase and insert at a different place in the text
- Press the **DELETE WORD** or **DELETE LINE** soft key: The text is removed and stored temporarily
- Move the cursor to the location where you wish insert the text, and press the **INSERT LINE / WORD** soft key

Soft key	Function
DELETE LINE	Delete and temporarily store a line
DELETE WORD	Delete and temporarily store a word
DELETE CHAR	Delete and temporarily store a character
INSERT LINE / WORD	Insert a line or word from temporary storage

Editing text blocks

You can copy and erase text blocks of any size, and insert them at other locations. Before any of these actions, you must first select the desired text block:

- To select a text block: Move the cursor to the first character of the text you wish to select.

SELECT BLOCK

- Press the **SELECT BLOCK** soft key
- Move the cursor to the last character of the text you wish to select. You can select whole lines by moving the cursor up or down directly with the arrow keys—the selected text is shown in a different color.

After selecting the desired text block, you can edit the text with the following soft keys:

Soft key	Function
CUT OUT BLOCK	Delete the selected block and store temporarily
COPY BLOCK	Store the selected block temporarily without erasing (copy)

If desired, you can now insert the temporarily stored block at a different location:

- Move the cursor to the location where you want to insert the temporarily stored text block

INSERT BLOCK

- Press the **INSERT BLOCK** soft key—the text block is inserted

You can insert the temporarily stored text block as often as desired

Transferring the selected block to a different file

- Select the text block as described previously

APPEND TO FILE

- Press the **APPEND TO FILE** soft key.
- > The control displays the **Destination file =** dialog message.
- Enter the path and the name of the destination file.
- > The control appends the selected text block to the specified file. If no target file with the specified name is found, the control creates a new file with the selected text.

Inserting another file at the cursor position

- Move the cursor to the location in the text where you wish to insert another file

READ FILE

- Press the **READ FILE** soft key.
- > The control displays the **File name =** dialog message.
- Enter the path and name of the file you want to insert

Finding text sections

With the text editor, you can search for words or character strings in a text. The control provides the following two options.

Finding the current text

The search function is used for finding the next occurrence of the word in which the cursor is presently located:

- Move the cursor to the desired word.
- To select the search function, press the **FIND** soft key
- Press the **FIND CURRENT WORD** soft key
- Find a word: Press the **FIND** soft key
- Exit the search function: Press the **END** soft key

Finding any text

- To select the search function, press the **FIND** soft key. The control displays the dialog prompt **Find text :**
- Enter the text that you wish to find
- Find text: Press the **FIND** soft key
- Exit the search function: Press the **END** soft key

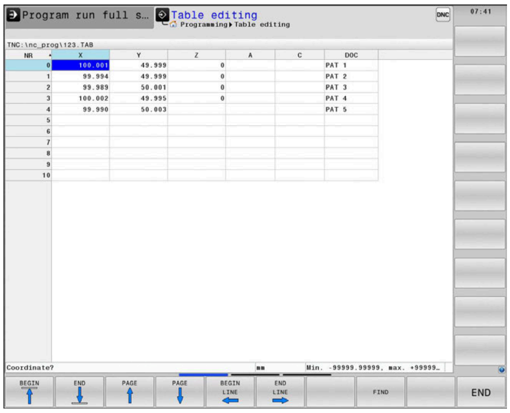
10.11 Freely definable tables

Fundamentals

In freely definable tables you can save and read any information from the NC program. The Q parameter functions **FN 26** to **FN 28** are provided for this purpose.

You can change the format of freely definable tables, i.e. the columns and their properties, by using the structure editor. They enable you to make tables that are exactly tailored to your application.

You can also toggle between a table view (standard setting) and form view.



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

Creating a freely definable table

Proceed as follows:

PGM MGT

- Press the **PGM MGT** key
- Enter any desired file name with the extension .TAB

ENT

- Confirm with the **ENT** key
- > The TNC displays a pop-up window with permanently saved table formats.
- Use the arrow key to select a table template, e.g. **example.tab**

ENT

- Confirm with the **ENT** key
- > The control opens a new table in the predefined format.
- To adapt the table to your requirements you have to edit the table format
  Further information: "Editing the table format", Page 388

> Refer to your machine manual!
>
> Machine tool builders may define their own table templates and save them in the control. When you create a new table, the control opens a pop-up window listing all available table templates.

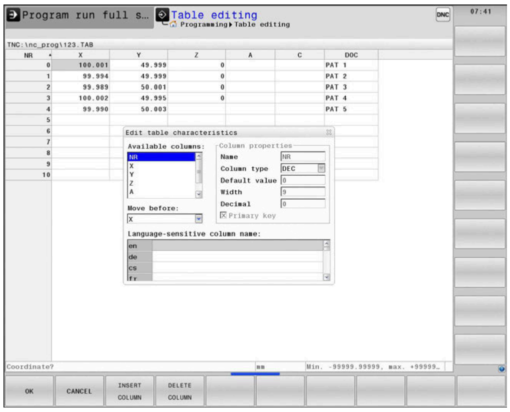
> You can also save your own table templates in the TNC. To do so, create a new table, change the table format and save the table in the **TNC:\system\proto** directory. If you then create new table, the control offers your template in the selection window for table templates.

Editing the table format

Proceed as follows:

EDIT FORMAT

- ▶ Press the **EDIT FORMAT** soft key
- > The control opens a pop-up window displaying the table structure.
- ▶ Adapt the format



The control provides the following options:

Structure command	Meaning
Available columns:	List of all columns contained in the table
Move before:	The entry highlighted in **Available columns** is moved in front of this column
Name	Column name: Is displayed in the header
Column type	**TEXT**: Text entry **SIGN**: + or - sign **BIN**: Binary number **DEC**: Decimal, positive, whole number (cardinal number) **HEX**: Hexadecimal number **INT**: Whole number **LENGTH**: Length (is converted in inch programs) **FEED**: Feed rate (mm/min or 0.1 inch/min) **IFEED**: Feed rate (mm/min or inch/min) **FLOAT**: Floating-point number **BOOL**: Logical value **INDEX**: Index **TSTAMP**: Fixed format for date and time **UPTEXT**: Text entry in upper case **PATHNAME**: Path name
Default value	Default value for the fields in this column
Width	Width of the column (number of charac-ters)
Primary key	First table column
Language-sensitive column name	Language-sensitive dialogs

Columns with a column type that permits letters, such as **TEXT**, can only be output or written to via QS parameters, even if the content of the cell is a number.

You can use a connected mouse or the navigation keys to move through the form.

Proceed as follows:

- Press the navigation keys to jump to the input fields

GOTO
- Press the **GOTO** key in order to open expandable menus

- Use the arrow keys to navigate within an input field

> In a table that already contains lines you can not change the table properties **Name** and **Column type**. Once you have deleted all lines, you can change these properties. If required, create a backup copy of the table beforehand.
>
> With the **CE** and **ENT** key combination, you can reset invalid values in fields with the **TSTAMP** column type.

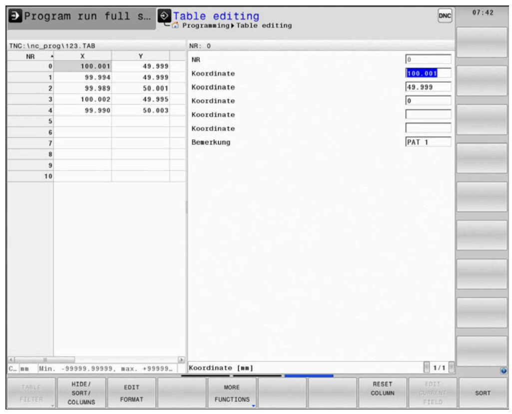
Close the structure editor

Proceed as follows:

OK
- Press the **OK** soft key
- > The control closes the editing form and applies the changes.

CANCEL
- Alternative: Press the **CANCEL** soft key
- > The control discards all entered changes.

Switching between table and form view



All tables with the **.TAB** extension can be opened in either list view or form view.

Switch the view as follows:

- Press the **Screen layout** key

FORM
- Press the soft key with the desired view

In the left half of the form view, the control lists the line numbers with the contents of the first column.

You can change the data as follows in the form view:

ENT
- Press the **ENT** key in order to switch to the next input field on the right-hand side

Selecting another row to be edited:

- Press the **Next tab** key
- > The cursor jumps to the left window.
- Use the arrow keys to select the desired row
- Press the **Next tab** key to switch back to the input window

FN 26: TABOPEN – Open a freely definable table

With the function **FN 26: TABOPEN** you open a freely definable table to be written to with **FN 27** or to be read from with **FN 28**.

> Only one table can be opened in an NC program at any one time. A new NC block with **FN 26: TABOPEN** automatically closes the last opened table.
>
> The table to be opened must have the extension **.TAB**.

Example: Open the table TAB1.TAB, which is saved in the directory TNC:\DIR1.

```
56 FN 26: TABOPEN TNC:\DIR1\TAB1.TAB
```

FN 27: TABWRITE – Write to a freely definable table

With the **FN 27: TABWRITE** function you write to the table that you previously opened with **FN 26: TABOPEN**.

You can define multiple column names in a **TABWRITE** block. The column names must be written between quotation marks and separated by a comma. You define in Q parameters the value that the control is to write to the respective column.

> The **FN 27: TABWRITE** function is considered only in the **Program run, single block** and **Program run, full sequence** operating modes.
>
> The **FN 18 ID992 NR16** function allows you to query the operating mode in which the NC program is running.
>
> If you write to more than one column in an NC block, you must save the values under successive Q parameter numbers.
>
> The control displays an error message if you try to write to a table cell that is locked or does not exist.

Use QS parameters if you want to write to a text field (such as column type **UPTEXT**). Use Q, QL, or QR parameters to write to numerical fields.

Example

You wish to write to the columns “Radius”, “Depth”, and “D” in line 5 of the presently opened table. The values to be written in the table are saved in the Q parameters **Q5**, **Q6**, and **Q7**.

```
53 Q5 = 3.75
54 Q6 = -5
55 Q7 = 7.5
56 FN 27:  TABWRITE 5/"RADIUS,DEPTH,D" = Q5
```

FN 28: TABREAD – Read from a freely definable table

With the **FN 28: TABREAD** function you read from the table previously opened with **FN 26: TABOPEN**.

You can define, i.e. read, multiple column names in a **TABREAD** block. The column names must be written between quotation marks and separated by a comma. In the **FN 28** block you can define the Q parameter number in which the control is to write the value that is first read.

> If you wish to read from more than one column in an NC block, the control will save the values under successive Q parameters of the same time, such as **QL1**, **QL2**, and **QL3**.

Use QS parameters if you want to read a text field. Use Q, QL, or QR parameters to read from numerical fields.

Example

You wish to read the values of the columns **X**, **Y**, and **D** from line 6 of the presently opened table. Save the first value in the Q parameter **Q10**, the second in **Q11**, and the third value in **Q12**.

From the same row, save the column **DOC** in **QS1**.

```
56 FN 28:  TABREAD Q10 = 6/“X,Y,D“
57 FN 28:  TABREAD QS1 = 6/“DOC“
```

Adapting the table format

NOTICE

Caution: Data may be lost!

The **ADAPT NC PGM / TABLE** function changes the format of all tables permanently. The control does not perform an automatic backup prior to a format change. The files will thus be permanently changed and may no longer be usable.

- Only use the function in consultation with the machine tool builder.

Soft key	Function
ADAPT NC PGM / TABLE	Adapt format of tables present after changing the control software version

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

10.12 Pulsing spindle speed FUNCTION S-PULSE

Programming a pulsing spindle speed

Application

Refer to your machine manual!

Read and note the functional description of the machine tool builder.

Follow the safety precautions.

Using the **S-PULSE FUNCTION** you can program a pulsing spindle speed, e.g. to avoid natural oscillations of the machine when operating at a constant spindle speed.

You can define the duration of a vibration (period length) using the P-TIME input value or a speed change in percent using the SCALE input value. The spindle speed changes in a sinusoidal form around the target value.

Procedure

Example

```
13 FUNCTION S-PULSE P-TIME10 SCALE5
```

Proceed as follows for the definition:

SPEC FCT
- Show the soft-key row with special functions

PROGRAM FUNCTIONS
- Press the **PROGRAM FUNCTIONS** soft key

FUNCTION SPINDLE
- Press the **FUNCTION SPINDLE** soft key

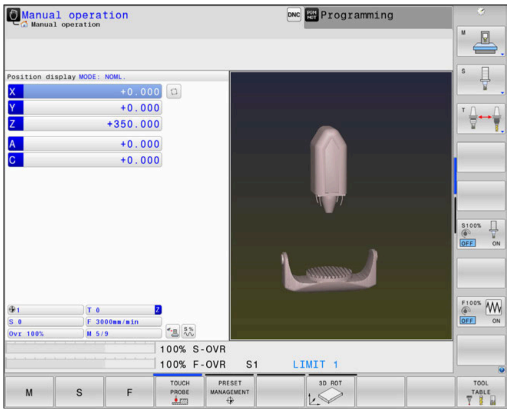
SPINDLE-PULSE
- Press the **SPINDLE-PULSE** soft key
- Define period length P-TIME
- Define speed change SCALE

The control never exceeds a programmed speed limit. The spindle speed is maintained until the sinusoidal curve of the **S-PULSE FUNCTION** falls below the maximum speed once more.

Symbols

In the status bar, the icon indicates the condition of the pulsing shaft speed:

Icon	Function
S %	Pulsing spindle speed active



Resetting the pulsing spindle speed

Example

18 FUNCTION S-PULSE RESET

Use the **FUNCTION S-PULSE RESET** to reset the pulsing spindle speed.

Proceed as follows for the definition:

SPEC FCT
- Show the soft-key row with special functions

PROGRAM FUNCTIONS
- Press the **PROGRAM FUNCTIONS** soft key

FUNCTION SPINDLE
- Press the **FUNCTION SPINDLE** soft key

RESET SPINDLE-PULSE
- Press the **RESET SPINDLE-PULSE** soft key.

10.13 Dwell time FUNCTION FEED

Programming dwell time

Application

Refer to your machine manual!

Read and note the functional description of the machine tool builder.

Follow the safety precautions.

The **FUNCTION FEED DWELL** function can be used to program a recurring dwell time in seconds, e.g. to force chip breaking in a turning cycle. Program **FUNCTION FEED DWELL** immediately prior to the machining you wish to run with chip breaking.

The defined dwell time from **FUNCTION FEED DWELL** is effective in both milling and turning operations.

The **FUNCTION FEED DWELL** function is not effective with rapid traverse movements and probing motion.

NOTICE

Caution: Danger to the tool and workpiece!

When the **FUNCTION FEED DWELL** function is active, the control will repeatedly interrupt the feed movement. While the feed movement is interrupted, the tool remains at its current position, and the spindle continues to turn. During thread cutting, this behavior will cause the workpiece to become scrap. There is also a risk of tool breakage during execution!

- Deactivate the **FUNCTION FEED DWELL** function before cutting threads

Procedure

Example

```
13 FUNCTION FEED DWELL D-TIME0.5 F-TIME5
```

Proceed as follows for the definition:

SPEC FCT
- Show the soft-key row with special functions

PROGRAM FUNCTIONS
- Press the **PROGRAM FUNCTIONS** soft key

FUNCTION FEED
- Press the **FUNCTION FEED** soft key

FEED DWELL
- Press the **FEED DWELL** soft key
- Define the interval duration for dwelling D-TIME
- Define the interval duration for cutting F-TIME

Resetting dwell time

> Reset to the dwell time immediately following the machining with chip breaking.

Example

```
18 FUNCTION FEED DWELL RESET
```

Use **FUNCTION FEED DWELL RESET** to reset the recurring dwell time.

Proceed as follows for the definition:

SPEC FCT
- Show the soft-key row with special functions

PROGRAM FUNCTIONS
- Press the **PROGRAM FUNCTIONS** soft key

FUNCTION FEED
- Press the **FUNCTION FEED** soft key

RESET FEED DWELL
- Press the **RESET FEED DWELL** soft key

> You can also reset the dwell time by entering D-TIME 0.
>
> The control automatically resets the **FUNCTION FEED DWELL** function at the end of a program.

10.14 Dwell time FUNCTION DWELL

Programming dwell time

Application

The **FUNCTION DWELL** function enables you to program a dwell time in seconds or define the number of spindle revolutions for dwelling.

The defined dwell time from **FUNCTION DWELL** is effective in both milling and turning operations.

Procedure

Example

```
13 FUNCTION DWELL TIME10
```

Example

```
23 FUNCTION DWELL REV5.8
```

Proceed as follows for the definition:

SPEC FCT
- Show the soft-key row with special functions

PROGRAM FUNCTIONS
- Press the **PROGRAM FUNCTIONS** soft key

FUNCTION DWELL
- **FUNCTION DWELL** soft key

DWELL TIME
- Press the **DWELL TIME** soft key
- Define the duration in seconds

DWELL REVOLUTIONS
- Alternatively, press the **DWELL REVOLUTIONS** soft key
- Define the number of spindle revolutions

10.15 Lift off tool at NC stop: FUNCTION LIFTOFF

Programming tool lift-off with FUNCTION LIFTOFF

Requirement

Refer to your machine manual!

This function must be configured and enabled by your machine tool builder. In the **CfgLiftOff** (no. 201400) machine parameter, the machine tool builder defines the path the control is to traverse for a **LIFTOFF** command. You can also use the **CfgLiftOff** machine parameter to deactivate the function.

In the **LIFTOFF** column of the tool table, set the **Y** parameter for the active tool.

Further information: User's Manual for Setup, Testing and Running NC Programs

Application

The **LIFTOFF** function is effective in the following situations:

- In case of an NC stop triggered by you
- In case of an NC stop triggered by the software, e. g. if an error has occurred in the drive system.
- In case of a power failure

The tool retracts from the contour by up to 2 mm. The control calculates the lift off direction based on the input in the **FUNCTION LIFTOFF** block.

You can program the **LIFTOFF** function in the following ways:

- **FUNCTION LIFTOFF TCS X Y Z**: Lift-off with a defined vector in the tool coordinate system
- **FUNCTION LIFTOFF ANGLE TCS SPB**: Lift-off with a defined angle in the tool coordinate system
- Lift-off in the tool axis direction with **M148**

Further information: "Automatically retracting the tool from the contour at an NC stop: M148", Page 240

Lift-off in turning mode

NOTICE

Caution: Danger to the tool and workpiece!

Undesired movements of the axes can occur if you use the **FUNCTION LIFTOFF ANGLE TCS** function in turning mode. The behavior of the control depends on the kinematics description and Cycle 800 (**Q498=1**).

- Carefully test the NC program or program section in the **Program run, single block** operating mode
- If necessary, change the algebraic sign of the defined angle

The control calculates the solution as follows:

- If the tool spindle is defined as an axis, the **LIFTOFF** will also rotate when reversing the tool.
- If the tool spindle is defined as a kinematic transformation, the **LIFTOFF** will **not** rotate when reversing the tool!

Further information: Cycle Programming User's Manual

Programming tool lift-off with a defined vector

Example

```
18 FUNCTION LIFTOFF TCS X+0 Y+0.5 Z+0.5
```

With **LIFTOFF TCS X Y Z**, you define the lift-off direction as a vector in the tool coordinate system. The control calculates the lift-off height in each axis based on the tool path defined by the machine tool builder.

Proceed as follows for the definition:

SPEC FCT

- Show the soft-key row with special functions

PROGRAM FUNCTIONS

- Press the **PROGRAM FUNCTIONS** soft key

FUNCTION LIFTOFF

- Press the **FUNCTION LIFTOFF** soft key

LIFTOFF TCS

- Press the **LIFTOFF TCS** soft key
- Enter X, Y, and Z vector components

Programming tool lift-off with a defined angle

Example

```
18 FUNCTION LIFTOFF ANGLE TCS SPB+20
```

With **LIFTOFF ANGLE TCS SPB**, you define the lift-off direction as a spatial angle in the tool coordinate system. This function is particularly helpful for turning operations.

The SPB angle you enter describes the angle between Z and X. If you enter 0°, the tool lifts off in the tool Z axis direction.

Proceed as follows for the definition:

SPEC FCT
- Show the soft-key row with special functions

PROGRAM FUNCTIONS
- Press the **PROGRAM FUNCTIONS** soft key

FUNCTION LIFTOFF
- Press the **FUNCTION LIFTOFF** soft key

LIFTOFF ANGLE TCS
- Press the **LIFTOFF ANGLE TCS** soft key
- Enter the SPB angle

Resetting the lift-off function

Example

```
18 FUNCTION LIFTOFF RESET
```

Use the **FUNCTION LIFTOFF RESET** to reset the lift-off function.

Proceed as follows for the definition:

SPEC FCT
- Show the soft-key row with special functions

PROGRAM FUNCTIONS
- Press the **PROGRAM FUNCTIONS** soft key

FUNCTION LIFTOFF
- Press the **FUNCTION LIFTOFF** soft key

LIFTOFF RESET
- Press the **LIFTOFF RESET** soft key

> You can also reset the lift-off with M149.
>
> The control automatically resets the **FUNCTION LIFTOFF** function at the end of a program.

11

Multiple-Axis-Machining

11.1 Functions for multiple axis machining

This chapter summarizes the control functions for multiple axis machining:

Control function	Description	Page
PLANE	Define machining in the tilted working plane	403
M116	Feed rate of rotary axes	435
PLANE/M128	Inclined-tool machining	433
FUNCTION TCPM	Define the behavior of the control when positioning the rotary axes (enhancement of M128)	443
M126	Shortest-path traverse of rotary axes	436
M94	Reduce display value of rotary axes	437
M128	Define the behavior of the control when positioning the rotary axes	438
M138	Selection of tilted axes	441
M144	Calculate machine kinematics	442
LN blocks	Three-dimensional tool compensation	449

11.2 The PLANE function: Tilting the working plane (option 8)

Introduction

Refer to your machine manual!

The machine manufacturer must enable the functions for tilting the working plane!

You can only use the **PLANE** function in its entirety on machines having at least two rotary axes (table axes, head axes or combined axes). The **PLANE AXIAL** function is an exception. **PLANE AXIAL** can also be used on machines which have only one programmed rotary axis.

The **PLANE** functions provide powerful options to define tilted working planes in various ways.

The parameter definition of the **PLANE** functions is subdivided into two parts:

- The geometric definition of the plane, which is different for each of the available **PLANE** functions.
- The positioning behavior of the **PLANE** function, which is independent of the plane definition and is identical for all **PLANE** functions
  Further information: "Defining the positioning behavior of the PLANE function", Page 422

NOTICE

Danger of collision!

When the machine is switched on, the control tries to restore the switch-off status of the tilted plane. This is prevented under certain conditions. For example, this applies if axis angles are used for tilting while the machine is configured with spatial angles, or if you have changed the kinematics.

- ▶ If possible, reset the tilted condition before switching the machine off
- ▶ Check the tilted condition when switching the machine back on

NOTICE

Danger of collision!

Cycle **8 MIRROR IMAGE** may have different effects in conjunction with the **Tilt working plane** function. The programming sequence, the mirrored axes, and the tilting function used are critical in this regard. There is a risk of collision during the tilting operation and subsequent machining!

- Check the sequence and positions using a graphic simulation
- Carefully test the NC program or program section in the **Program run, single block** operating mode

Examples

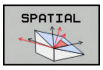
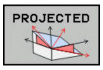
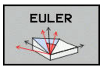
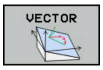
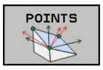
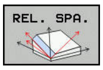
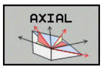
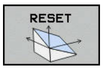
1. When Cycle **8 MIRROR IMAGE** is programmed before the tilting function without rotary axes:
   - The tilt of the **PLANE** function used (except **PLANE AXIAL**) is mirrored
   - The mirroring is effective after the tilt with **PLANE AXIAL** or Cycle **19**
2. When Cycle **8 MIRROR IMAGE** is programmed before the tilting function with a rotary axis:
   - The mirrored rotary axis has no effect on the tilt specified in the **PLANE** function used, because only the movement of the rotary axis is mirrored

Operating and programming notes:

- The actual-position-capture function is not possible with an active tilted working plane.
- If you use the **PLANE** function when **M120** is active, the control automatically rescinds the radius compensation, which also rescinds the **M120** function.
- Always use **PLANE RESET** to cancel **PLANE** functions. Entering 0 in all **PLANE** parameters (e.g. all three spatial angles) exclusively resets the angles, but not the function.
- If you restrict the number of tilting axes with the **M138** function, your machine may provide only limited tilting possibilities. The machine tool builder will decide whether the control takes the angles of deselected axes into account or sets them to 0.
- The control only supports tilting the working plane with spindle axis Z.

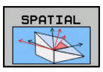
Overview

Most **PLANE** functions (except **PLANE AXIAL**) can be used to describe the desired working plane independently of the rotary axes available on your machine. The following possibilities are available:

Soft key	Function	Required parameters	Page
	**SPATIAL**	Three spatial angles: **SPA**, **SPB**, and **SPC**	408
	**PROJECTED**	Two projection angles: **PROPR** and **PROMIN** and a rotation angle **ROT**	410
	**EULER**	Three Euler angles: precession (**EULPR**), nutation (**EULNU**) and rotation (**EULROT**),	412
	**VECTOR**	Normal vector for defining the plane and base vector for defining the direction of the tilted X axis	414
	**POINTS**	Coordinates of any three points in the plane to be tilted	417
	**RELATIVE**	Single, incrementally effective spatial angle	419
	**AXIAL**	Up to three absolute or incremental axis angles **A,B,C**	420
	**RESET**	Reset the PLANE function	407

Running an animation

To familiarize yourself with the various definition possibilities of each **PLANE** function, you can start animated sequences via soft key. To do so, first enter animation mode and then select the desired **PLANE** function. While the animation plays, the control highlights the soft key of the selected **PLANE** function with a blue color.

Soft key	Function
	Switch on the animation mode
	Select the desired animation (highlighted in blue)

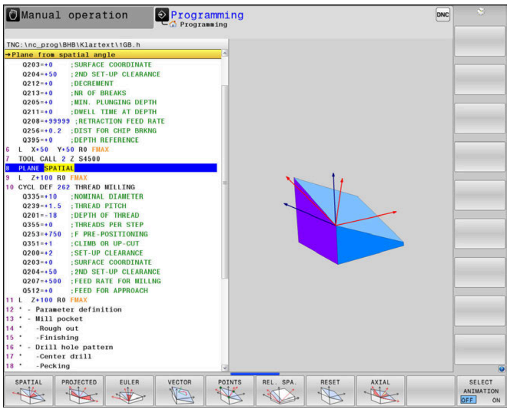
Defining the PLANE function

SPEC FCT

- Show the soft-key row with special functions

TILT MACHINING PLANE

- Press the **TILT MACHINING PLANE** soft key
- > The control display the available **PLANE** functions in the soft-key row.
- Select the **PLANE** function



Selecting functions

- Press the soft key linked to the desired function
- > The control continues the dialog and prompts you for the required parameters.

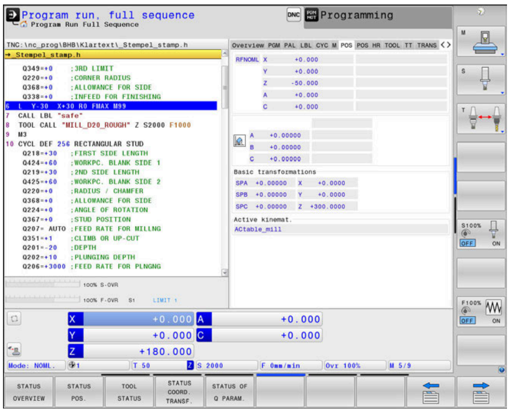
Selecting the function while animation is active

- Press the soft key linked to the desired function
- > The control plays the animation.
- To apply the currently active function, press the soft key of that function again or press the **ENT** key

Position display

As soon as a **PLANE** function (except **PLANE AXIAL**) is active, the control shows the calculated spatial angle in the additional status display.

During tilting into position (**MOVE** or **TURN** mode), the control shows, in the rotary axis, the distance to go to the calculated final position of the rotary axis in the distance-to-go display (**ACTDST** and **REFDST**).



Resetting PLANE function

Example

```
25 PLANE RESET MOVE DIST50 F1000
```

SPEC FCT

- Show the soft-key row with special functions

TILT MACHINING PLANE

- Press the **TILT MACHINING PLANE** soft key
- > The control displays the available **PLANE** functions in the soft-key row

RESET

- Select the reset function

MOVE

- Specify whether the control should automatically move the tilting axes to home position (**MOVE** or **TURN**) or not (**STAY**)
  Further information: "Automatic tilting into position MOVE/TURN/STAY", Page 423

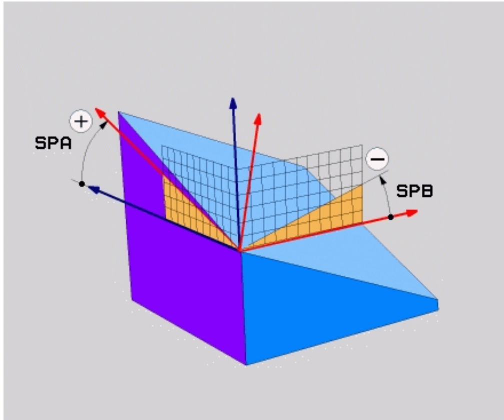
END

- Press the **END** key.

> The **PLANE RESET** function resets the active tilt and the angles (**PLANE** function or Cycle **19**) (angle = 0 and function inactive). It does not need to be defined more than once.
>
> You deactivate tilting via the 3D ROT menu in **Manual operation** mode.
>
> **Further information:** User's Manual for Setup, Testing and Running NC Programs

Defining the working plane with the spatial angle: PLANE SPATIAL

Application



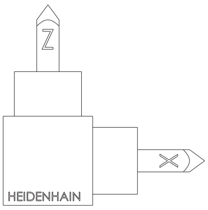
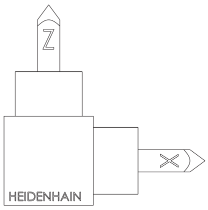
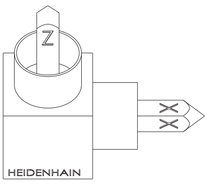
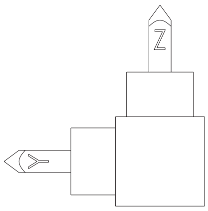
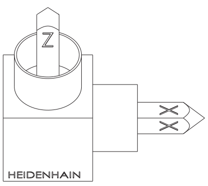
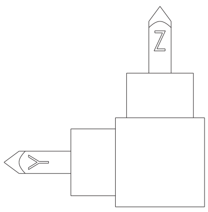
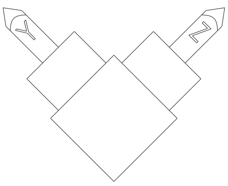
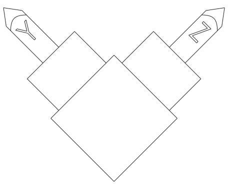
Spatial angles define a working plane through up to three rotations in the non-tilted workpiece coordinate system (**tilting sequence A-B-C**).

Most users assume three successive rotations in reverse order (**tilting sequence C-B-A**).

The result is identical for both perspectives, as the following comparison shows.

Example

```
PLANE SPATIAL SPA+45 SPB+0 SPC+90 ...
```

A-B-C	C-B-A
Home position A0° B0° C0° 	Home position A0° B0° C0° 
A+45° 	C+90° 
B+0° 	B+0° 
C+90° 	A+45° 

Comparison of the tilting orders:

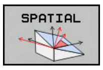
- **Tilting order A-B-C:**
  1. Tilt about the non-tilted X axis of the workpiece coordinate system
  2. Tilt about the non-tilted Y axis of the workpiece coordinate system
  3. Tilt about the non-tilted Z axis of the workpiece coordinate system
- **Tilting order C-B-A:**
  1. Tilt about the non-tilted Z axis of the workpiece coordinate system
  2. Tilt about the tilted Y axis
  3. Tilt about the tilted X axis

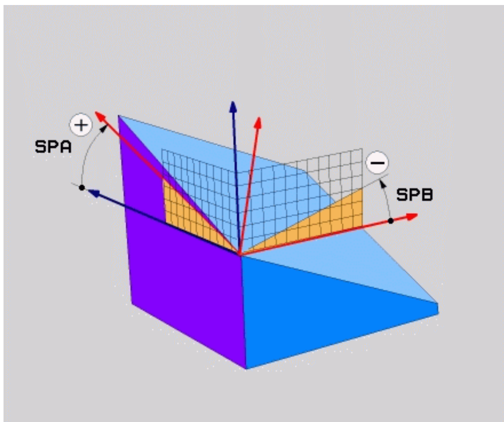
> Programming notes:
> - You must always define all three spatial angles **SPA**, **SPB** and **SPC**, even if one or more have the value 0.
> - Depending on the machine, Cycle **19** requires you to enter spatial angles or axis angles. If the configuration (machine parameter setting) allows the input of spatial angles, the angle definition is the same in Cycle **19** and in the **PLANE SPATIAL** function.
> - You can select the desired positioning behavior. **Further information:** "Defining the positioning behavior of the PLANE function", Page 422

Input parameters

Example

```
5 PLANE  SPATIAL SPA+27  SPB+0 SPC+45  .....
```

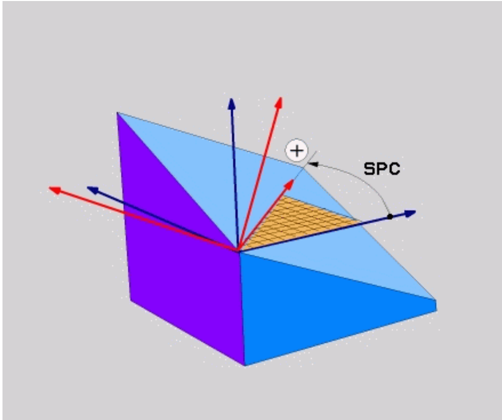




- **Spatial angle A?**: Rotational angle **SPA** about the (non-tilted) X axis. Input range from -359.9999 to +359.9999
- **Spatial angle B?**: Rotational angle **SPB** about the (non-tilted) Y axis. Input range from -359.9999 to +359.9999
- **Spatial angle C?**: Rotational angle **SPC** about the (non-tilted) Z axis. Input range from -359.9999 to +359.9999
- Continue with the positioning properties **Further information:** "Defining the positioning behavior of the PLANE function", Page 422

Abbreviations used

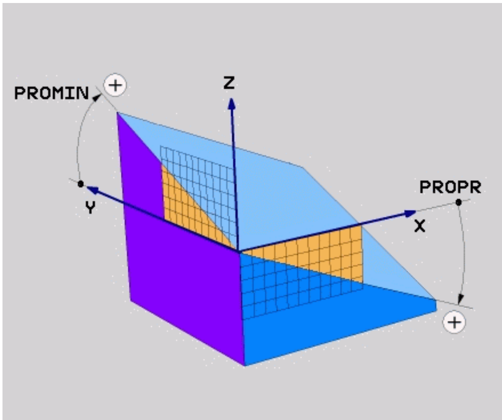
Abbreviation	Meaning
SPATIAL	In space
SPA	**Sp**atial **A**: Rotation about the (non-tilted) X axis
SPB	**Sp**atial **B**: Rotation about the (non-tilted) Y axis
SPC	**Sp**atial **C**: Rotation about the (non-tilted) Z axis



Defining the working plane with the projection angle: PLANE PROJECTED

Application

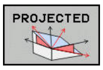
Projection angles define a working plane by specifying two angles that you can communicate by projection of the 1st coordinate plane (Z/X on tool axis Z) and 2nd coordinate plane (Y/Z on tool axis Z) to the working levels to be defined.

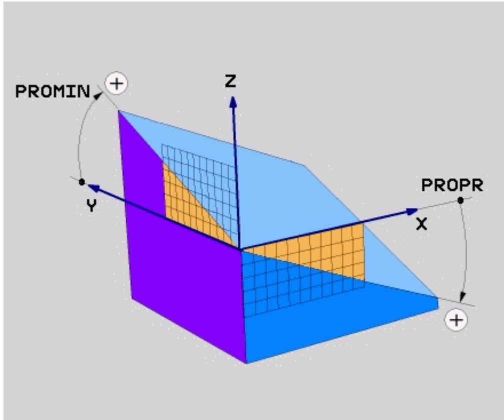


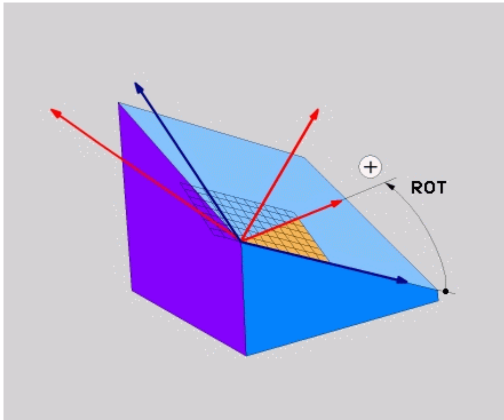
Programming notes:

- The projection angles correspond to the angle projections on the planes of a rectangular coordinate system. The angles at the outer faces of the workpiece only are identical to the projection angles if the workpiece is rectangular. Thus, with workpieces that are not rectangular, the angle specifications from the engineering drawing often differ from the actual projection angles.
- You can select the desired positioning behavior.
  Further information: "Defining the positioning behavior of the PLANE function", Page 422

Input parameters







- **Projection angle on 1st Coordinate plane?**: Projected angle of the tilted machining plane in the 1st coordinate plane of the untilted coordinate system (Z/X for tool axis Z). Input range: from –89.9999° to +89.9999°. The 0° axis is the principal axis of the active working plane (X for tool axis Z, positive direction)
- **Proj. angle on 2nd Coordinate plane?**: Projected angle in the 2nd coordinate plane of the untilted coordinate system (Y/Z for tool axis Z). Input range: from –89.9999° to +89.9999°. The 0° axis is the minor axis of the active machining plane (Y for tool axis Z)
- **ROT angle of tilted plane?**: Rotation of the tilted coordinate system around the tilted tool axis (corresponds to a rotation with Cycle 10 ROTATION). The rotation angle is used to simply specify the direction of the principal axis of the working plane (X for tool axis Z, Z for tool axis Y). Input range: –360° to +360°
- Continue with the positioning properties **Further information:** "Defining the positioning behavior of the PLANE function", Page 422

Example

```
5 PLANE PROJECTED PROPR+24 PROMIN+24 ROT+30 .....
```

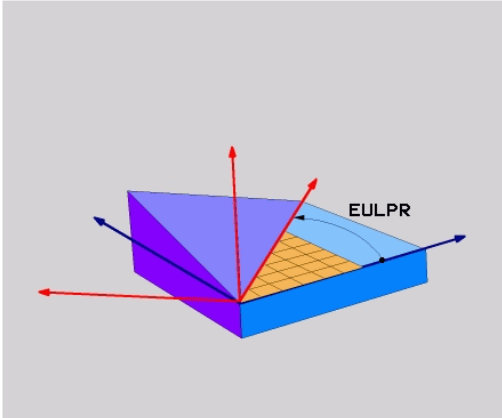
Abbreviations used:

PROJECTED	Projected
PROPR	Principal plane
PROMIN	Minor plane
ROT	Rotation

Defining the working plane with the Euler angle: PLANE EULER

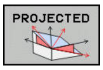
Application

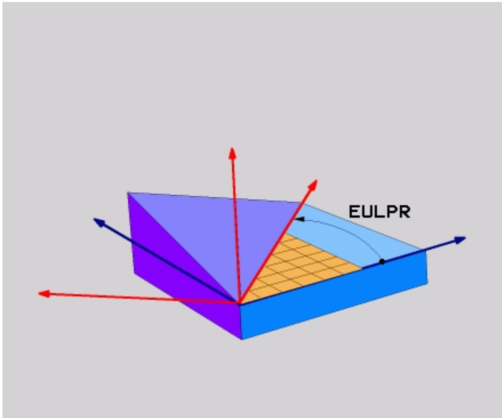
Euler angles define a machining plane through up to three **rotations about the respectively tilted coordinate system**. The Swiss mathematician Leonhard Euler defined these angles.



> You can select the desired positioning behavior.
> **Further information:** "Defining the positioning behavior of the PLANE function", Page 422

Input parameters





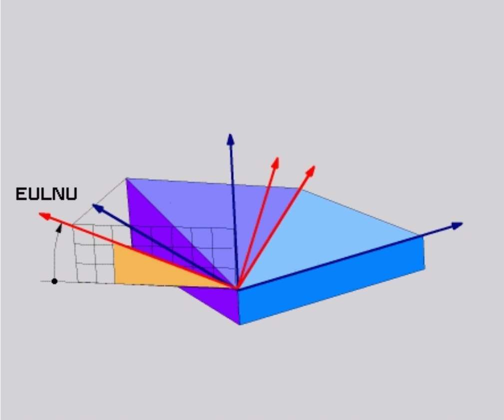
- **Rot. angle Main coordinate plane?:** Rotary angle **EULPR** around the Z axis. Please note:
  - Input range: -180.0000° to 180.0000°
  - The 0° axis is the X axis
- **Tilting angle tool axis?**: Tilting angle **EULNUT** of the coordinate system around the X axis shifted by the precession angle. Please note:
  - Input range: 0° to 180.0000°
  - The 0° axis is the Z axis
- **ROT angle of tilted plane?**: Rotation **EULROT** of the tilted coordinate system around the tilted Z axis (corresponds to a rotation with Cycle 10 ROTATION). Use the rotation angle to simply define the direction of the X axis on the tilted working plane. Please note:
  - Input range: 0° to 360.0000°
  - The 0° axis is the X axis
- Continue with the positioning properties
  Further information: "Defining the positioning behavior of the PLANE function", Page 422

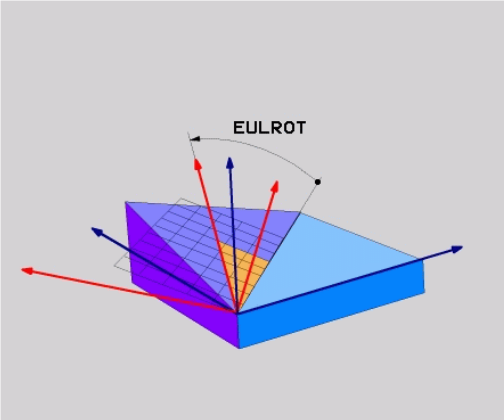
Example

```
5 PLANE  EULER EULPR45  EULNU20 EULROT22  .....
```

Abbreviations used

Abbreviation	Meaning
EULER	Swiss mathematician who defined these angles
EULPR	**Pr**ecession angle: angle describing the rotation of the coordinate system around the Z axis
EULNU	**Nu**tation angle: angle describing the rotation of the coordinate system around the X axis shifted by the precession angle
EULROT	**Rot**ation angle: angle describing the rotation of the tilted machining plane around the tilted Z axis



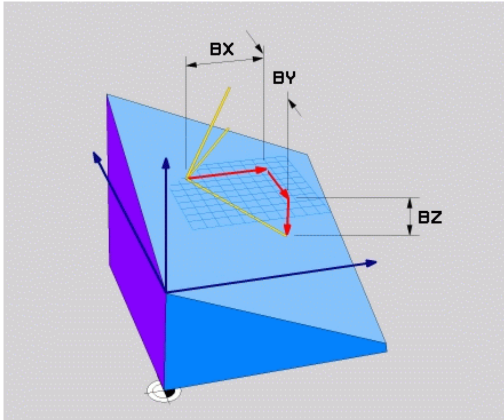


Defining the working plane with two vectors: PLANE VECTOR

Application

You can use the definition of a working plane via **two vectors** if your CAD system can calculate the base vector and normal vector of the tilted machining plane. A normalized input is not necessary. The control internally calculates the normal, so you can enter values between -9.999999 and +9.999999.

The base vector required for the definition of the machining plane is defined by the components **BX, BY** and **BZ.** The normal vector is defined by the components **NX**, **NY** and **NZ**.



Programming notes:

- The control calculates standardized vectors from the values you enter.
- The normal vector defines the slope and the orientation of the working plane. The base vector defines the orientation of the main axis X in the defined working plane. To ensure that the definition of the working plane is unambiguous, you must program the vectors perpendicular to each other. The machine tool builder defines how the control will behave for vectors that are not perpendicular.
- The programmed normal vector must not be too short, e.g. all directional components having a length of 0 or 0.0000001. In this case, the control would not be able to determine the slope. Machining is aborted and an error message is displayed. This behavior is independent of the configuration of the machine parameters.
- You can select the desired positioning behavior. **Further information:** "Defining the positioning behavior of the PLANE function", Page 422

Refer to your machine manual!

The machine tool builder configures the behavior of the control with vectors that are not perpendicular.

Alternatively to generating the default error message, the control can correct (or replace) the base vector that is not perpendicular. This correction (or replacement) does not affect the normal vector.

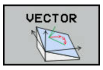
Default correction behavior of the control if the base vector is not perpendicular:

- The base vector is projected along the normal vector onto the working plane (defined by the normal vector).

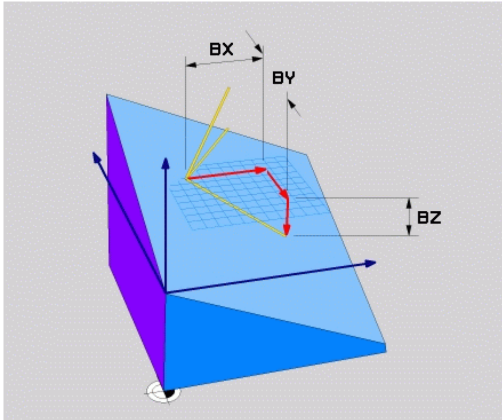
Correction behavior of the control if the base vector is not perpendicular and too short, parallel or antiparallel to the normal vector:

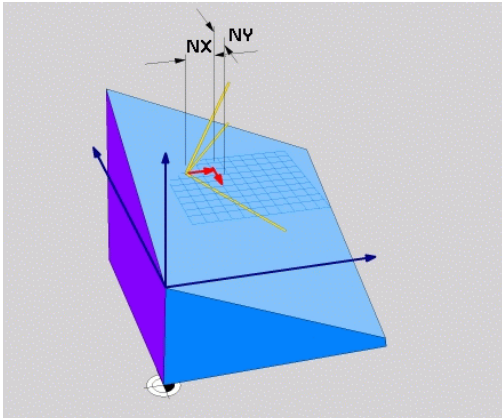
- If the normal vector has no X component, the base vector corresponds to the original X axis
- If the normal vector has no Y component, the base vector corresponds to the original Y axis

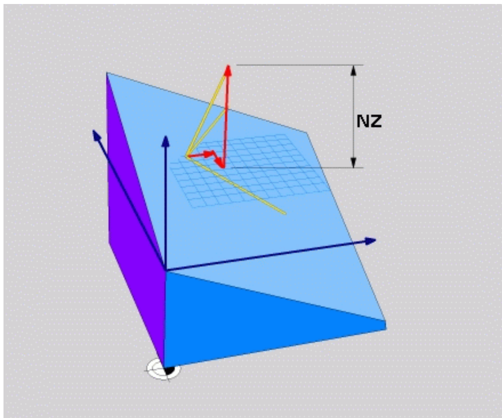
Input parameters



- **X component of base vector?**: X component **BX** of the base vector B; input range: from -9.9999999 to +9.9999999
- **Y component of base vector?**: Y component **BY** of the base vector B; input range: from -9.9999999 to +9.9999999
- **Z component of base vector?**: Z component **BZ** of the base vector B; input range: from -9.9999999 to +9.9999999
- **X component of normal vector?**: X component **NX** of the normal vector N; input range: from -9.9999999 to +9.9999999
- **Y component of normal vector?**: Y component **NY** of the normal vector N; input range: from -9.9999999 to +9.9999999
- **Z component of normal vector?**: Z component **NZ** of the normal vector N; input range: from -9.9999999 to +9.9999999
- Continue with the positioning properties
  Further information: "Defining the positioning behavior of the PLANE function", Page 422







Example

```
5 PLANE VECTOR BX0.8 BY-0.4 BZ-0.42 NX0.2 NY0.2 NZ0.92 ..
```

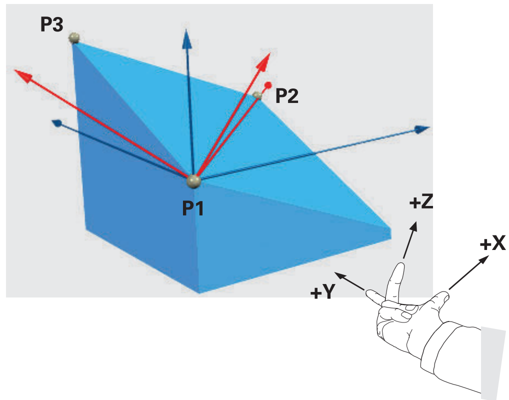
Abbreviations used

Abbreviation	Meaning
VECTOR	Vector
BX, BY, BZ	**B**ase vector : **X**, **Y**, and **Z** components
NX, NY, NZ	**N**ormal vector : **X**, **Y**, and **Z** components

Defining the working plane via three points: PLANE POINTS

Application

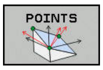
A working plane can be uniquely defined by entering **any three points P1 to P3 in this plane**. This possibility is realized in the **PLANE POINTS** function.



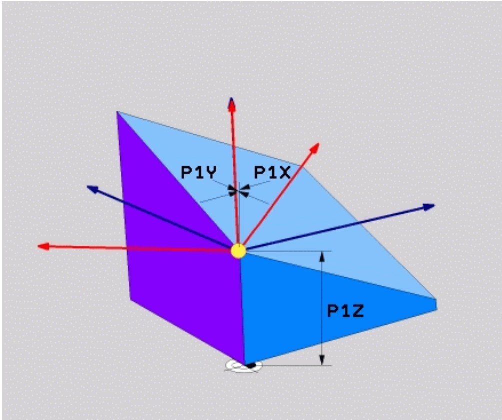
Programming notes:

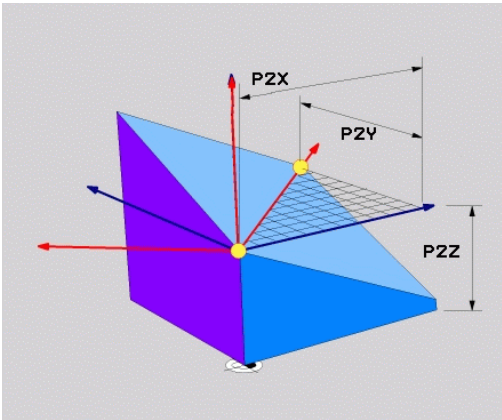
- The three points define the slope and orientation of the plane. The position of the active datum is not changed through **PLANE POINTS**.
- Point 1 and Point 2 determine the orientation of the tilted main axis X (for tool axis Z).
- Point 3 defines the slope of the tilted working plane. In the defined working plane, the Y axis is automatically oriented perpendicularly to the main axis X. The position of Point 3 thus also determines the orientation of the tool axis and consequently the orientation of the working plane. To have the positive tool axis pointing away from the workpiece, Point 3 must be located above the connection line between Point 1 and Point 2 (right-hand rule).
- You can select the desired positioning behavior.
  Further information: "Defining the positioning behavior of the PLANE function", Page 422

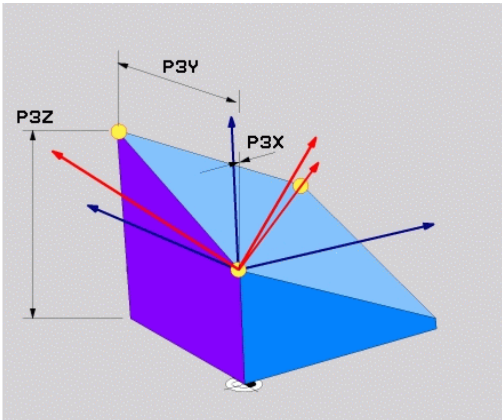
Input parameters



- **X coordinate of 1stplane point?**: X coordinate **P1X** of the 1st plane point
- **Y coordinate of 1stplane point?**: Y coordinate **P1Y** of the 1st plane point
- **Z coordinate of 1stplane point**: Z coordinate **P1Z** of the 1st plane point
- **X coordinate of 2ndplane point?**: X coordinate **P2X** of the 2nd plane point
- **Y coordinate of 2ndplane point?**: Y coordinate **P2Y** of the 2nd plane point
- **Z coordinate of 2ndplane point?**: Z coordinate **P2Z** of the 2nd plane point
- **X coordinate of 3rdplane point?**: X coordinate **P3X** of the 3rd plane point
- **Y coordinate of 3rdplane point?**: Y coordinate **P3Y** of the 3rd plane point
- **Z coordinate of 3rdplane point?**: Z coordinate **P3Z** of the 3rd plane point
- Continue with the positioning properties
  Further information: "Defining the positioning behavior of the PLANE function", Page 422







Example

```
5 PLANE  POINTS P1X+0  P1Y+0 P1Z+20  P2X+30 P2Y+31  P2Z+20
   P3X+0 P3Y+41  P3Z+32.5 .....
```

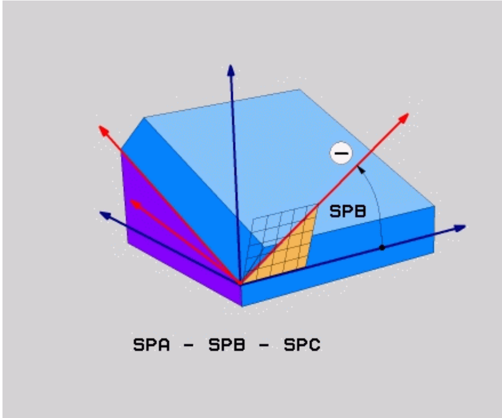
Abbreviations used

Abbreviation	Meaning
POINTS	Points

Defining the working plane via a single incremental spatial angle: PLANE RELATIV

Application

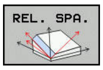
Use a relative spatial angle when an already active tilted working plane is to be tilted by **another rotation**. Example: machining a 45° chamfer on a tilted plane.



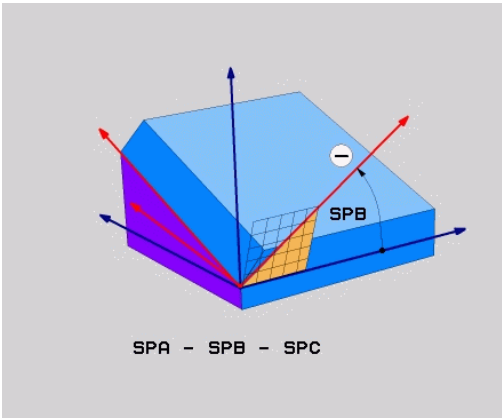
Programming notes:

- The defined angle is always in effect in respect to the active working plane, regardless of the tilting function you used before.
- You can program any number of **PLANE RELATIV** functions in a row..
- If you want to return the working plane to the orientation that was active before the **PLANE RELATIV** function, define the same **PLANE RELATIV** function again but enter the value with the opposite algebraic sign.
- If you use **PLANE RELATIV** without previous tilting, **PLANE RELATIV** will be effective directly in the workpiece coordinate system. In this case, you can tilt the original working plane by entering a defined spatial angle in the **PLANE RELATIV** function.
- You can select the desired positioning behavior. **Further information:** "Defining the positioning behavior of the PLANE function", Page 422

Input parameters



- **Incremental angle?**: Spatial angle by which the active machining plane is to be rotated. Use a soft key to select the axis to be rotated around. Input range: -359.9999° to +359.9999°
- Continue with the positioning properties **Further information:** "Defining the positioning behavior of the PLANE function", Page 422



Example

```
5 PLANE  RELATIV SPB-45  .....
```

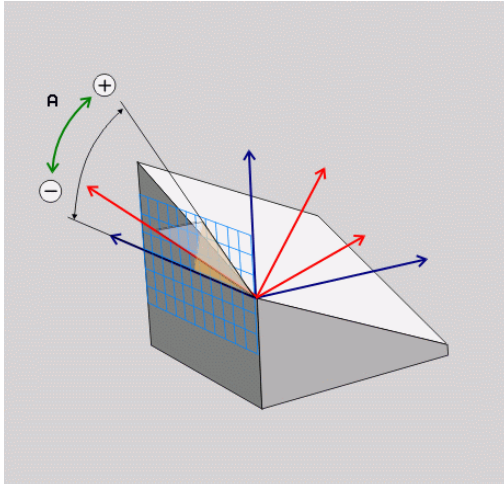
Abbreviations used

Abbreviation	Meaning
RELATIVE	Relative to

Tilting the working plane through axis angle: PLANE AXIAL

Application

The **PLANE AXIAL** function defines both the slope and the orientation of the working plane and the nominal coordinates of the rotary axes.



PLANE AXIAL can also be used on machines that have only one rotary axis.

The input of nominal coordinates (axis angle input) is advantageous in that it provides an unambiguously defined tilting situation based on defined axis positions. Spatial angles entered without an additional definition are often mathematically ambiguous. Without the use of a CAM system, entering axis angles, in most cases, only makes sense if the rotary axes are positioned perpendicularly.

Refer to your machine manual!

If your machine allows spatial angle definitions, you can continue your programming with **PLANE RELATIV** after **PLANE AXIAL**.

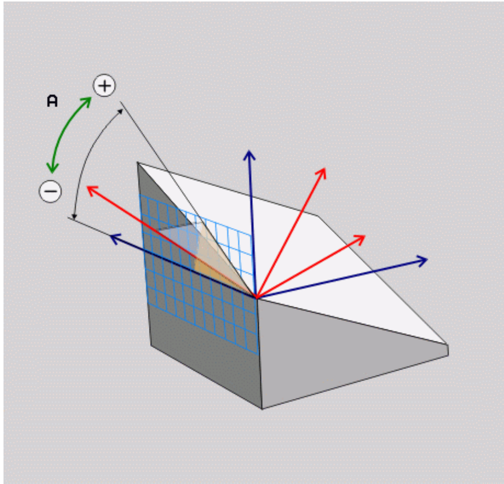
Programming notes:

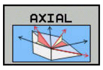
- The axis angles must correspond to the axes present on the machine. If you try to program axis angles for rotary axes that do not exist on the machine, the control will generate an error message.
- Use **PLANE RESET** to reset the **PLANE AXIAL** function. Entering 0 only resets the axis angle, but does not deactivate the tilting function.
- The axis angles of the **PLANE AXIAL** function are modally effective. If you program an incremental axis angle, the control will add this value to the currently effective axis angle. If you program two different rotary axes in two successive **PLANE AXIAL** functions, the new working plane is derived from the two defined axis angles.
- **SYM** (**SEQ**), **TABLE ROT**, and **COORD ROT** have no function in conjunction with **PLANE AXIAL**.
- The **PLANE AXIAL** function does not take basic rotation into account.

Input parameters

Example

```
5 PLANE AXIAL B-45 .....
```





- **Axis angle A?**: Axis angle **to which** the A axis is to be tilted. If entered incrementally, it is the angle **by which** the A axis is to be tilted from its current position. Input range: –99999.9999° to +99999.9999°
- **Axis angle B?**: Axis angle **to which** the B axis is to be tilted. If entered incrementally, it is the angle **by which** the B axis is to be tilted from its current position. Input range: –99999.9999° to +99999.9999°
- **Axis angle C?**: Axis angle **to which** the C axis is to be tilted. If entered incrementally, it is the angle **by which** the C axis is to be tilted from its current position. Input range: –99999.9999° to +99999.9999°
- Continue with the positioning properties **Further information:** "Defining the positioning behavior of the PLANE function", Page 422

Abbreviations used

Abbreviation	Meaning
AXIAL	In the axial direction

Defining the positioning behavior of the PLANE function

Overview

Independently of which PLANE function you use to define the tilted machining plane, the following functions are always available for the positioning behavior:

- Automatic positioning
- Selecting alternate tilting options (not for **PLANE AXIAL**)
- Selecting the type of transformation (not for **PLANE AXIAL**)

NOTICE

Danger of collision!

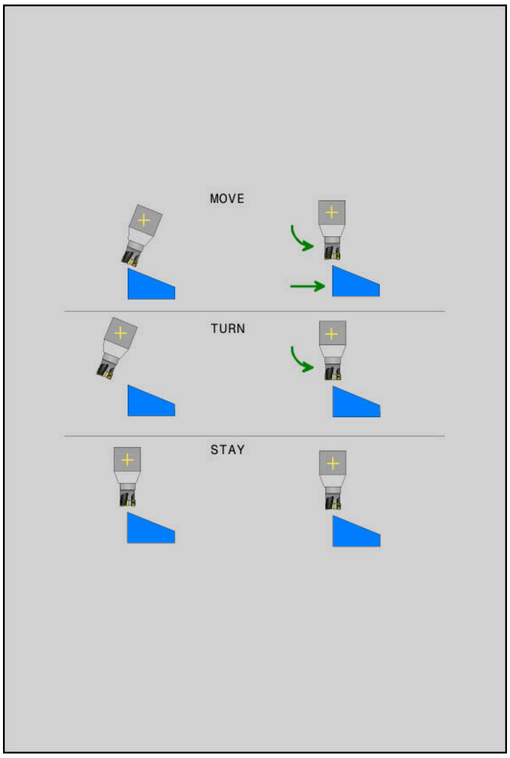
Cycle **8 MIRROR IMAGE** may have different effects in conjunction with the **Tilt working plane** function. The programming sequence, the mirrored axes, and the tilting function used are critical in this regard. There is a risk of collision during the tilting operation and subsequent machining!

- ▶ Check the sequence and positions using a graphic simulation
- ▶ Carefully test the NC program or program section in the **Program run, single block** operating mode

Examples

1. When Cycle **8 MIRROR IMAGE** is programmed before the tilting function without rotary axes:
   - The tilt of the **PLANE** function used (except **PLANE AXIAL**) is mirrored
   - The mirroring is effective after the tilt with **PLANE AXIAL** or Cycle **19**
2. When Cycle **8 MIRROR IMAGE** is programmed before the tilting function with a rotary axis:
   - The mirrored rotary axis has no effect on the tilt specified in the **PLANE** function used, because only the movement of the rotary axis is mirrored

Automatic tilting into position MOVE/TURN/STAY



After you have entered all of the parameters for the plane definition, you must specify how the control is to tilt the rotary axes to the calculated axis value. This entry is mandatory.

The control offers the following ways of tilting the rotary axes to the calculated axis values:

- The PLANE function is to automatically tilt the rotary axes to the calculated axis values, with the relative position between the tool and the workpiece remaining the same.
- > The control carries out a compensating movement in the linear axes.

- The PLANE function is to automatically tilt the rotary axes to the calculated axis values, during which only the rotary axes are positioned.
- > The control does **not** carry out a compensating movement in the linear axes.

- You tilt the rotary axes into position in a subsequent, separate positioning block

If you have selected the **MOVE** option (**PLANE** function is to automatically tilt into position with a compensation movement), then the two subsequently explained parameters **Dist. tool tip - center of rot.** and **Feed rate? F=** must still be defined.

If you have selected the **TURN** option (**PLANE** function is to automatically tilt into position without compensation movement), then the subsequently explained **Feed rate?** parameter **F=** must still be defined.

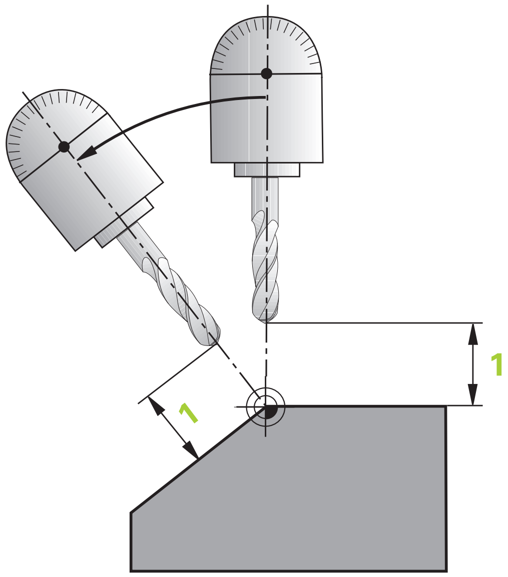
As an alternative to a feed rate **F** defined directly by a numerical value, you can also tilt the axes into position with **FMAX** (rapid traverse) or **FAUTO** (feed rate from the **TOOL CALL** block).

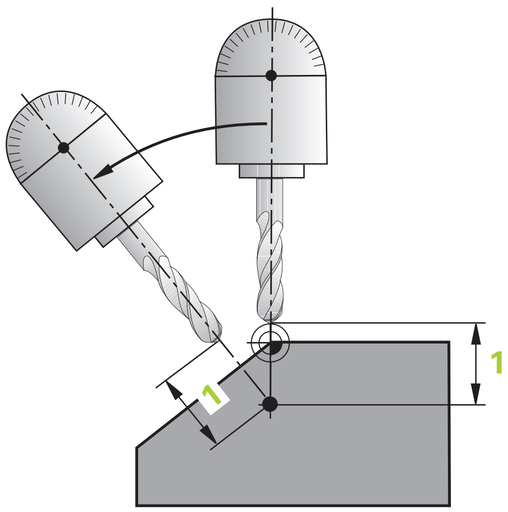
> If you use **PLANE** together with **STAY,** you have to position the rotary axes in a separate block after the **PLANE** function.

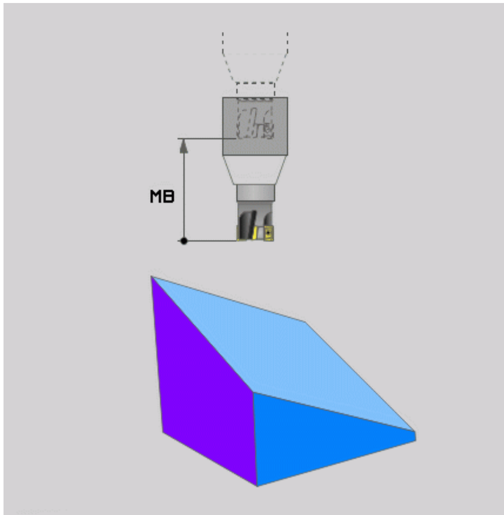
- **Dist. tool tip - center of rot.** (incremental): The **DIST** parameter shifts the center of rotation of the tilting movement relative to the current position of the tool tip.
  - If the tool is already at the specified distance from the workpiece prior to being tilted into position, then it will be at the same relative position after being tilted into position (see center figure on the right, **1** = DIST)
  - If the tool is not at the specified distance from the workpiece before being tilted into position, then it will be offset relative to the original position after being tilted into position (see lower figure on the right, **1** = DIST)

> The control tilts the tool (or table) relative to the tool tip.

- **Feed rate? F=**: Contour speed at which the tool is to be tilted into position
- **Retraction length in the tool axis?**: The retraction path **MB** takes effect incrementally from the current tool position in the active tool axis direction that the control approaches **before tilting**. **MB MAX** moves the tool to a position just before the software limit switch







Tilting the rotary axes into position in a separate NC block

To tilt the rotary axes into position in a separate positioning block (**STAY** option selected), proceed as follows:

NOTICE

Danger of collision!

The control does not automatically check whether collisions can occur between the tool and the workpiece. Incorrect or no pre-positioning before tilting the tool into position can lead to a risk of collision during the tilting movement!

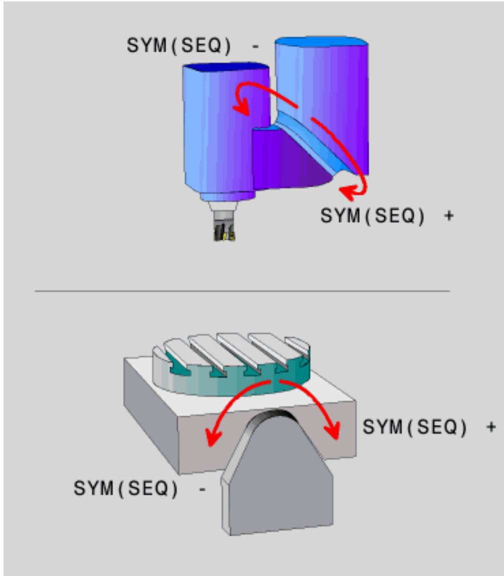
- Program a safe position before the tilting movement
- Carefully test the NC program or program section in the **Program run, single block** operating mode

- Select any **PLANE** function, and define automatic tilting into position with **STAY**. During program execution, the control calculates the position values of the rotary axes present on the machine and stores them in the system parameters Q120 (A axis), Q121 (B axis) and Q122 (C axis)
- Define the positioning block with the angular values calculated by the control

Example: Tilt a machine with a rotary table C and a tilting table A to a spatial angle of B+45

...	
12 L Z+250 R0 FMAX	Position at clearance height
13 PLANE SPATIAL SPA+0 SPB+45 SPC+0 STAY	Define and activate the PLANE function
14 L A+Q120 C+Q122 F2000	Position the rotary axis with the values calculated by the control.
...	Define machining in the tilted working plane

Selection of tilting possibilities SYM (SEQ) +/–



Based on the position that you have defined for the working plane, the control must calculate the appropriate position of the rotary axes present on your machine. In general, there are always two possible solutions.

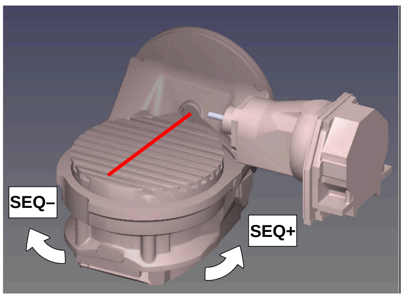
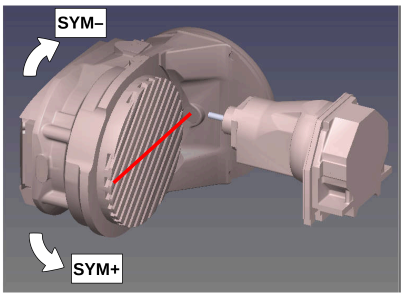
For the selection of one of the possible solutions, the control offers two variants: **SYM** and **SEQ**. You use soft keys to choose the variants. **SYM** is the standard variant.

The entry of **SYM** or **SEQ** is optional.

SEQ assumes that the master axis is in its home position (0°). Relative to the tool, the master axis is the first rotary axis or the last rotary axis relative to the table (depending on the machine configuration). If both possible solutions are in the positive or negative range, then the control automatically uses the closer solution (shorter path). If you need the second possible solution, then you must either pre-position the master axis (in the area of the second possible solution) before tilting the working plane, or work with **SYM**.

Unlike **SEQ**, **SYM** uses the symmetry point of the master axis as its reference. Every master axis has two symmetry positions, which are 180° apart from each other (sometimes only one symmetry position is in the traverse range).

> Determine the symmetry point in the following manner:
> - Perform **PLANE SPATIAL** with any spatial angle and **SYM+**
> - Save the axis angle of the master axis in a Q parameter, (e.g., –100)
> - Repeat the **PLANE SPATIAL** function with **SYM-**
> - Save the axis angle of the master axis in a Q parameter (e.g., –80)
> - Calculate the average value (e.g., –90)
>
> The average value corresponds to the symmetry point.

Reference for SEQ	Reference for SYM
	

With the **SYM** function, you select one of the possible solutions relative the symmetry point of the master axis:

- **SYM+** positions the master axis in the positive half-space relative to the symmetry point
- **SYM-** positions the master axis in the negative half-space relative to the symmetry point

With the **SEQ** function, you select one of the possible solutions relative to the home position of the master axis:

- **SEQ+** positions the master axis in the positive tilting range relative to home position
- **SEQ-** positions the master axis in the negative tilting range relative to home position

If the solution you have selected with **SYM** (**SEQ**) is not within the machine's range of traverse, then the control displays the **Entered angle not permitted** error message.

> If the **PLANE AXIAL** function is used, the **SYM** (**SEQ**) function has no effect.

If you do not define **SYM** (**SEQ**), then the control determines the solution as follows:

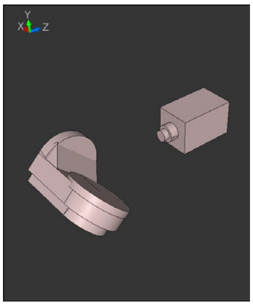
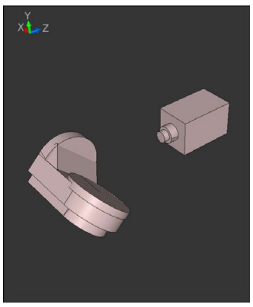
1. Check whether both possible solutions are within the traverse range of the rotary axes
2. Two possible solutions: Based on the current position of the rotary axes, choose the possible solution with the shortest path
3. One possible solution: Choose the only solution
4. No possible solution: Issue the error message **Entered angle not permitted**

Examples

Machine with C rotary axis and A tilting table. Programmed function: PLANE SPATIAL SPA+0 SPB+45 SPC+0

Limit switch	Start position	SYM = SEQ	Resulting axis position
None	A+0, C+0	Not prog.	A+45, C+90
None	A+0, C+0	+	A+45, C+90
None	A+0, C+0	–	A–45, C–90
None	A+0, C–105	Not prog.	A–45, C–90
None	A+0, C–105	+	A+45, C+90
None	A+0, C–105	–	A–45, C–90
–90 < A < +10	A+0, C+0	Not prog.	A–45, C–90
–90 < A < +10	A+0, C+0	+	Error message
–90 < A < +10	A+0, C+0	–	A–45, C–90

Machine with B rotary axis and A tilting table (limit switches: A +180 and –100). Programmed function: PLANE SPATIAL SPA-45 SPB+0 SPC+0

SYM	SEQ	Resulting axis position	Kinematics view
+		A–45, B+0	
-		Error message	**No solution in limited range**
	+	Error message	**No solution in limited range**
	-	A–45, B+0	

The position of the symmetry point is contingent on the kinematics. If you change the kinematics (such as switching the head), then the position of the symmetry point changes as well.

Depending on the kinematics, the positive direction of rotation of **SYM** may not correspond to the positive direction of rotation of **SEQ**. Therefore, ascertain the position of the symmetry point and the direction of rotation of **SYM** on each machine before programming.

Selection of the transformation type

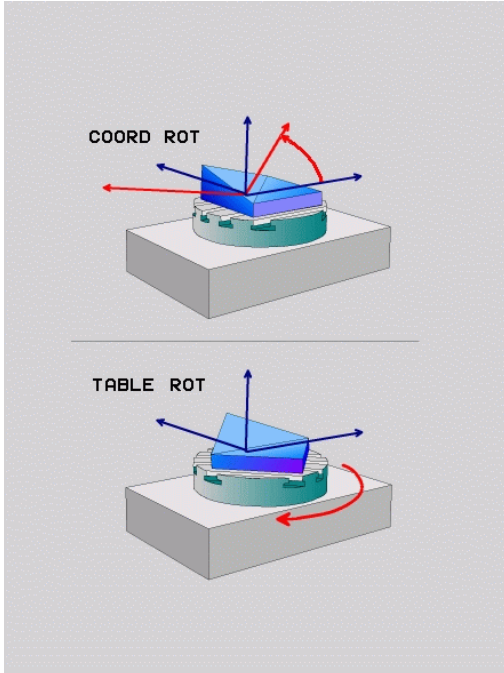
The **COORD ROT** and **TABLE ROT** transformation types influence the orientation of the working plane coordinate system through the axis position of a so-called free rotary axis.

The entry of **COORD ROT** or **TABLE ROT** is optional.

Any rotary axis becomes a free rotary axis with the following configuration:

- The rotary axis has no effect on the tool angle of inclination because the rotary axis and the tool axis are parallel in the tilting situation
- The rotary axis is the first rotary axis in the kinematic chain starting from the workpiece

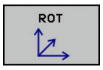
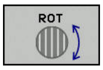
The effect of the **COORD ROT** and **TABLE ROT** transformation types therefore depends on the programmed spatial angles and the machine kinematics.



> Programming notes:
> - If no free rotary axis arises in a tilting situation, then the **COORD ROT** and **TABLE ROT** transformation types have no effect.
> - With the **PLANE AXIAL** function, the **COORD ROT** and **TABLE ROT** transformation types have no effect.

Effect with a free rotary axis

> Programming notes
> - For the positioning behavior with the **COORD ROT** and **TABLE ROT** transformation types, it makes no difference whether the free rotary axis is a table axis or a head axis.
> - The resulting axis position of the free rotary axis depends on an active basic rotation, among other factors.
> - The orientation of the working plane coordinate system also depends on a programmed rotation; for example, by means of Cycle 10 **ROTATION**.

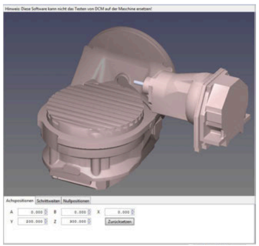
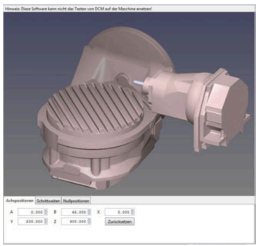
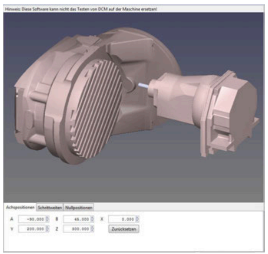
Soft key	Effect
	**COORD ROT**: > The control positions the free rotary axis to 0 > The control orients the working plane coordinate system in accordance with the programmed spatial angle
	**TABLE ROT** with: ■ SPA **and** SPB **equal to** 0 ■ SPC **equal or unequal to** 0 > The control orients the free rotary axis in accordance with the programmed spatial angle > The control orients the working plane coordinate system in accordance with the basic coordinate system **TABLE ROT** with: ■ **At least** SPA **or** SPB **unequal to** 0 ■ SPC **equal or unequal to** 0 > The control does not position the free rotary axis. The position prior to tilting the working plane is maintained > Since the workpiece was not positioned, the control orients the working plane coordinate system in accordance with the programmed spatial angle

> If no transformation type was selected, then the control uses the **COORD ROT** transformation type for the **PLANE** functions

Example

The following example shows the effect of the **TABLE ROT** transformation type in conjunction with a free rotary axis.

...	
6 L B+45 R0 FMAX	Pre-position rotary axis
7 PLANE SPATIAL SPA-90 SPB+20 SPC+0 TURN F5000 TABLE ROT	Tilt the working plane
...	

Origin	A = 0, B = 45	A = –90, B = 45
		

- The control positions the B axis to the axis angle B+45
- With the programmed tilting situation with SPA–90, the B axis becomes the free rotary axis
- The control does not position the free rotary axis. The position of the B axis prior to the tilting of the working plane is maintained
- Since the workpiece was not also positioned, the control orients the working plane coordinate system in accordance with the programmed spatial angle SPB+20

Tilting the working plane without rotary axes

> Refer to your machine manual!
>
> This function must be enabled and adapted by the machine tool builder.
>
> The machine tool builder must take the precise angle into account, e.g. the angle of a mounted angular head in the kinematics description.

You can also orient the programmed working plane perpendicularly to the tool without defining rotary axes, e.g. when adapting the working plane for a mounted angular head.

Use the **PLANE SPATIAL** function and the **STAY** positioning behavior to swivel the working plane to the angle specified by the machine tool builder.

Example of mounted angular head with permanent tool direction Y:

Example

```
TOOL CALL 5 Z S4500
PLANE SPATIAL SPA+0 SPB-90 SPC+0 STAY
```

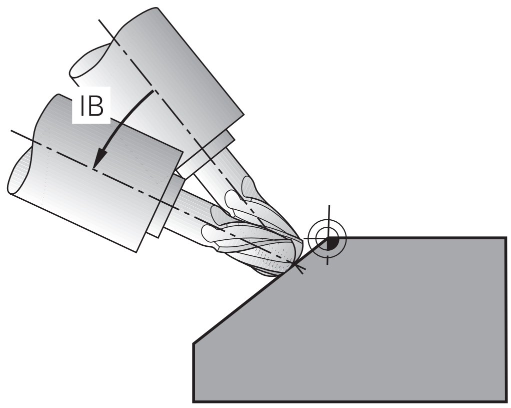
> The tilt angle must be precisely adapted to the tool angle, otherwise the control will generate an error message.

11.3 Inclined-tool machining in a tilted plane (option 9)

Function

In combination with **M128** and the new **PLANE** functions, **inclined-tool machining** on a tilted machining plane is now possible. Two possibilities are available for definition:

- Inclined-tool machining via incremental traverse of a rotary axis
- Inclined-tool machining via normal vectors



> Inclined-tool machining in a tilted machining plane only works with spherical cutters. If you are using 45° swivel heads and tilting tables, you can also define the incline angle as a spatial angle. Use **FUNCTION TCPM** for this purpose.
>
> **Further information:** "FUNCTION TCPM (option 9)", Page 443

Inclined-tool machining via incremental traverse of a rotary axis

- Retract the tool
- Define any PLANE function; consider the positioning behavior
- Activate M128
- Via a straight-line block, traverse to the desired incline angle in the appropriate axis incrementally

Example

...	
12 L Z+50 R0 FMAX	Position at clearance height
13 PLANE SPATIAL SPA+0 SPB-45 SPC+0 MOVE DIST50 F1000	Define and activate the PLANE function
14 M128	Activate M128
15 L IB-17 F1000	Set the incline angle
...	Define machining in the tilted working plane

Inclined-tool machining via normal vectors

> Only one directional vector can be defined in the **LN** block. This vector defines the incline angle (normal vector **NX**, **NY**, **NZ** or tool directional vector **TX**, **TY**, **TZ**).

- Retract the tool
- Define any PLANE function; consider the positioning behavior
- Activate M128
- Execute NC program with LN blocks in which the tool direction is defined by a vector

Example

...	
12 L Z+50 R0 FMAX	Position at clearance height
13 PLANE SPATIAL SPA+0 SPB+45 SPC+0 MOVE DIST50 F1000	Define and activate the PLANE function
14 M128	Activate M128
15 LN X+31.737 Y+21.954 Z+33.165 NX+0.3 NY+0 NZ +0.9539 F1000 M3	Set the incline angle with the normal vector
...	Define machining in the tilted working plane

11.4 Miscellaneous functions for rotary axes

Feed rate in mm/min on rotary axes A, B, C: M116 (option 8)

Standard behavior

The control interprets the programmed feed rate of a rotary axis in degrees/min (in mm programs and also in inch programs). The feed rate therefore depends on the distance from the tool center to the center of the rotary axis.

The larger this distance becomes, the greater the contouring feed rate.

Feed rate in mm/min on rotary axes with M116

Refer to your machine manual!

The machine geometry must be specified by the machine tool builder in the kinematics description.

Programming notes:

- The **M116** function can be used with table axes and head axes.
- The **M116** function also has an effect if the **Tilt working plane** function is active.
- It is not possible to combine the **M128** or **TCPM** function with **M116**. If you want to activate **M116** for an axis while the **M128** or **TCPM** function is active, then you must indirectly deactivate the compensating movement for this axis using **M138**. This is done indirectly because, with **M138**, you specify the axis for which the **M128** or **TCPM** function takes effect. Thus, **M116** automatically affects the axis that was not selected with **M138**.
  Further information: "Selecting tilting axes: M138", Page 441
- Without the **M128** or **TCPM** function, **M116** can take effect for two rotary axes at the same time.

The control interprets the programmed feed rate of a rotary axis in mm/min (or 1/10 inch/min). In this case, the control calculates the feed for the block at the start of each NC block. The feed rate of a rotary axis will not change while the NC block is executed, even if the tool moves toward the center of the rotary axis.

Effect

M116 is effective in the working plane. Reset **M116** with **M117**. At the end of the program, **M116** is automatically canceled.

M116 becomes effective at the start of the block.

Shorter-path traverse of rotary axes: M126

Standard behavior

Refer to your machine manual!

The positioning behavior of rotary axes is machine-dependent.

M126 has an effect only on modulo axes.

In the case of modulo axes, the axis position begins again at 0° after the modulo length of 0° – 360° has been exceeded. This is the case for rotary axes that are mechanically capable of endless rotation.

In the case of non-modulo axes, the maximum rotation is mechanically limited. The position display of the rotary axis does not switch back to the starting value (e.g., 0° – 540°).

The default behavior of the control for the positioning of rotary axes whose position display is reduced to a traversing range of less than 360° depends on the machine parameter **shortestDistance** (no. 300401) . This machine parameter defines whether the control will position along the shortest path between the nominal and actual position even without M126.

Behavior without M126:

Without **M126**, the control moves a rotary axis whose position display is reduced to less than 360° along a long path.

Examples:

Actual position	Nominal position	Range of traverse
350°	10°	–340°
10°	340°	+330°

Behavior with M126

With **M126**, the control moves a rotary axis whose position display is reduced to less than 360° on the shortest path of traverse.

Examples:

Actual position	Nominal position	Range of traverse
350°	10°	+20°
10°	340°	–30°

Effect

M126 takes effect at the start of the block.

M127 and a program end reset **M126**.

Reducing display of a rotary axis to a value less than 360°: M94

Standard behavior

The control moves the tool from the current angular value to the programmed angular value.

Example:

Current angular value:	538°
Programmed angular value:	180°
Actual distance of traverse:	-358°

Behavior with M94

At the start of block, the control first reduces the current angular value to a value less than 360° and then moves the tool to the programmed value. If multiple rotary axes are active, **M94** will reduce the display of all rotary axes. As an alternative, you can specify a rotary axis after **M94**. The control then reduces the display of this axis only.

If you entered a traverse limit or a software limit switch is active, **M94** is ineffective for the corresponding axis.

Example: Reduce the display of all active rotary axes

```
L M94
```

Example: Reduce the display of the C axis

```
L M94 C
```

Example: Reduce the display of all active rotary axes and then move the tool in the C axis to the programmed value

```
L C+180  FMAX M94
```

Effect

M94 is effective only in the NC block where it is programmed.

M94 becomes effective at the start of the block.

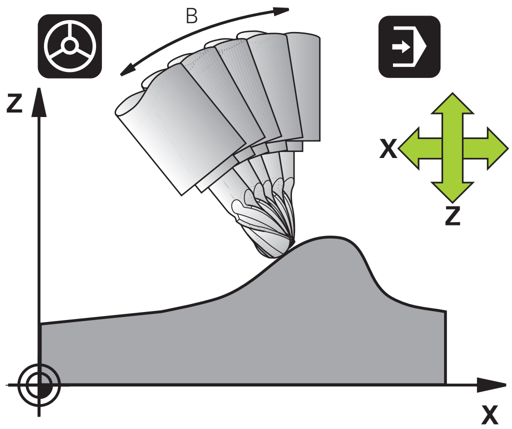
Retain position of the tool tip during the positioning of tilting axes (TCPM): M128 (option 9)

Standard behavior

If the inclination angle of the tool changes, this results in an offset of the tool tip compared to the nominal position. The control does not compensate for this offset. If the operator does not take this deviation into account in the NC program, offset machining is executed.

Behavior with M128 (TCPM: Tool Center Point Management)

If the position of a controlled tilting axis changes in the NC program, then the position of the tool tip relative to the workpiece remains unchanged.



NOTICE

Danger of collision!

Rotary axes with Hirth coupling must move out of the coupling to enable tilting. There is a danger of collision while the axis moves out of the coupling and during the tilting operation.

▶ Retract the tool before changing the position of the tilting axis

After **M128**, you can still enter a maximum feed rate at which the control will carry out the compensating movements in the linear axes.

If you want to change the position of the tilting axis with the handwheel during program run, then use **M128** in conjunction with **M118**. The superimposing of handwheel positioning is performed with active **M128**, depending on the setting in the 3D ROT menu of **Manual operation** mode, in the active coordinate system or in the non-tilted coordinate system.

Programming notes:

- Before positioning with **M91** or **M92**, and before a **TOOL CALL** block, reset the **M128** function
- To avoid contour damage, use only radius cutters with **M128**
- The tool length must be measured from the spherical center of the Ball-nose cutter
- If **M128** is active, then the control shows the **TCPM** symbol in the status display
- The **TCPM** or **M128** function cannot be used in conjunction with the **Dynamic Collision Monitoring (DCM)** function and the additional **M118** function

M128 on tilting tables

If you program a tilting table movement while **M128** is active, then the control rotates the coordinate system accordingly. For example, if you rotate the C axis by 90° (through a positioning or datum shift) and then program a movement in the X axis, then the control executes the movement in the machine Y axis.

The control also transforms the set preset, which has been shifted by the movement of the rotary table.

M128 with three-dimensional tool compensation

If you carry out a three-dimensional tool compensation with active **M128** and active radius compensation **RL**/**RR**, then the control will automatically position the rotary axes for certain machine geometries (peripheral milling).

Further information: "Three-dimensional tool compensation (option 9)", Page 449

Effect

M128 becomes effective at the start of the block, **M129** at the end of the block. **M128** also takes effect in the manual operating modes and remains active even after a change in the operating mode. The feed rate for the compensating movement remains in effect until you program a new feed rate or reset **M128** with **M129**.

You can reset **M128** with **M129**. The control also resets **M128** when you select a new NC program in a program run mode.

Example: Perform compensation movements at a feed rate of no more than 1000 mm/min

```
L X+0  Y+38.5 IB-15 RL  F125 M128  F1000
```

Inclined machining with non-controlled rotary axes

If your machine has non-controlled rotary axes (so-called counter axes), then you can also perform inclined machining operations with these axes in conjunction with **M128**

Proceed as follows:

1. Manually traverse the rotary axes to the desired positions. **M128** must not be active during this operation
2. Activate **M128**: the control reads the actual values of all existing rotary axes, calculates from this the new position of the tool center point, and updates the position display
3. The control performs the necessary compensating movement in the next positioning block
4. Execute the machining operation
5. At program end, reset **M128** with **M129**, and return the rotary axes to their initial positions

> As long as **M128** is active, the control monitors the actual positions of the non-controlled rotary axes. If the actual position deviates from the value that is definable by the machine manufacturer, then the control issues an error message and interrupts program run.

Selecting tilting axes: M138

Standard behavior

With the functions **M128**, **TCPM** and **Tilt working plane**, the control considers those rotary axes that have been specified by the machine manufacturer in the machine parameters.

Behavior with M138

The control performs the above functions only in those tilting axes that you have defined using **M138**.

> Refer to your machine manual!
>
> If you restrict the number of tilting axes with the **M138** function, your machine may provide only limited tilting possibilities. The machine tool builder will decide whether the control takes the angles of deselected axes into account or sets them to 0.

Effect

M138 becomes effective at the start of the block.

You can cancel **M138** by reprogramming it without specifying any axes.

Example

Perform the above-mentioned functions only in the tilting axis C.

```
L Z+100 R0 FMAX M138 C
```

Compensating the machine kinematics in ACTUAL/ NOMINAL positions at end of block: M144 (Option 9)

Standard behavior

If the kinematics change, e.g. by inserting an adapter spindle or entering an inclination angle, the control will not compensate this modification. If the operator does not consider this modification to the kinematics for the NC program, machining will occur with an offset.

Behavior with M144

> Refer to your machine manual!
> The machine geometry must be specified by the machine tool builder in the kinematics description.

The **M144** function enables the control to consider the modification to the machine kinematics in the position display and compensate the offset of the tool tip in relation to the workpiece.

> Programming and operating notes:
> - Positioning blocks with **M91** or **M92** are permitted while **M144** is active.
> - The position display in the **Program Run Full Sequence** and **Program Run Single Block** operating modes does not change until the tilting axes have reached their final position.

Effect

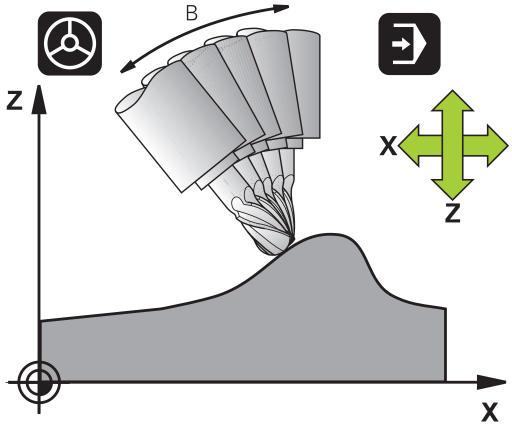
M144 becomes effective at the start of the block. **M144** does not work in connection with **M128** or the Tilt Working Plane function.

You can cancel **M144** by programming **M145**.

11.5 FUNCTION TCPM (option 9)

Function

Refer to your machine manual!
The machine geometry must be specified by the machine tool builder in the kinematics description.



FUNCTION TCPM is an improvement on the **M128** function, with which you can define the behavior of the control during the positioning of rotary axes. With **FUNCTION TCPM**, you can define the effects of various functions yourself:

- Effect of the programmed feed rate: **F TCP** / **F CONT**
- Interpretation of the rotary axis coordinates programmed in the NC program: **AXIS POS** / **AXIS SPAT**
- Type of orientation interpolation between the start and end positions: **PATHCTRL AXIS** / **PATHCTRL VECTOR**
- Optional selection of a tool reference point and a center of rotation: **REFPNT TIP-TIP** / **REFPNT TIP-CENTER** / **REFPNT CENTER-CENTER**
- Maximum feed rate at which the control performs the compensation movements in the linear axes: **F**

If **FUNCTION TCPM** is active, the control shows the **TCPM** symbol in the position display.

NOTICE

Danger of collision!

Rotary axes with Hirth coupling must move out of the coupling to enable tilting. There is a danger of collision while the axis moves out of the coupling and during the tilting operation.

- Retract the tool before changing the position of the tilting axis

Programming notes:

- Before positioning axes with **M91** or **M92,** and before a **TOOL CALL** block, reset the **FUNCTION TCPM** function.
- To avoid contour damage, use only Ball-nose cutters for face milling operations. In combination with other tool shapes, you should check the NC program for possible contour damage using the graphical simulation.

Defining FUNCTION TCPM

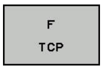
SPEC FCT
- Select the special functions

PROGRAM FUNCTIONS
- Select the programming aids

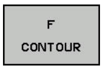
FUNCTION TCPM
- Select **FUNCTION TCPM**

Mode of action of the programmed feed rate

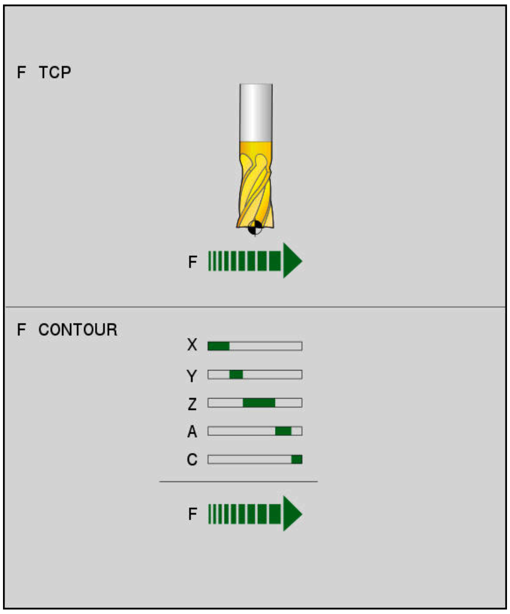
The control provides two functions for defining the operating method of the programmed feed rate:



- **F TCP** determines that the programmed feed rate is interpreted as the actual relative velocity between the tool tip (**t**ool **c**enter **p**oint) and the workpiece



- **F CONT** determines that the programmed feed rate is interpreted as the contouring feed rate of the axes programmed in the respective NC block.



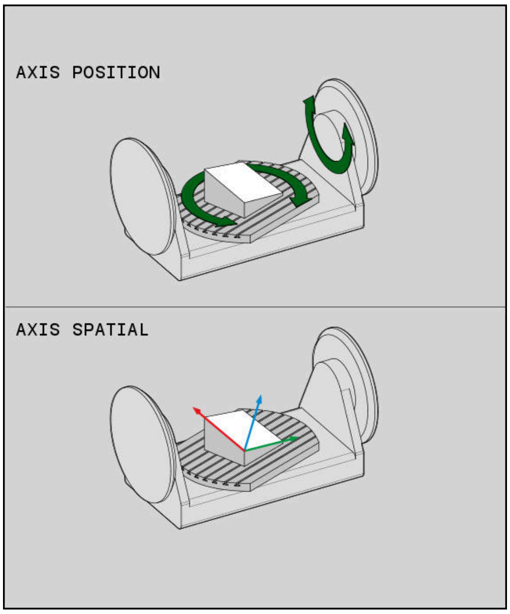
Example

...	
13 FUNCTION TCPM F TCP ...	Feed rate refers to the tool tip
14 FUNCTION TCPM F CONT ...	Feed rate is interpreted as the speed of the tool along the contour
...	

Interpretation of the programmed rotary axis coordinates

Up to now, machines with 45° swivel heads or 45° tilting tables could not easily set the angle of inclination or a tool orientation with respect to the currently active coordinate system (spatial angle). This function could only be realized through externally created NC programs with surface-normal vectors (LN blocks).

The control provides the following functionality:



AXIS POSITION

- **AXIS POS** determines that the control interprets the programmed coordinates of rotary axes as the nominal position of the respective axis

AXIS SPATIAL

- **AXIS SPAT** determines that the control interprets the programmed coordinates of rotary axes as spatial angles

Programming notes:

- The **AXIS POS** function is primarily suitable in conjunction with perpendicularly arrayed rotary axes. Only if the programmed rotary axis coordinates define the working plane correctly (e.g. programmed using a CAM system), you can also use **AXIS POS** with different machine designs (e.g. 45° swivel heads).
- The **AXIS SPAT** function is used to define spatial angles that are given with respect to the active coordinate system (which might be tilted). The defined angles have the effect of incremental spatial angles. Always program all three spatial angles in the first positioning block after the **AXIS SPAT** function, even if they are 0°.

Example

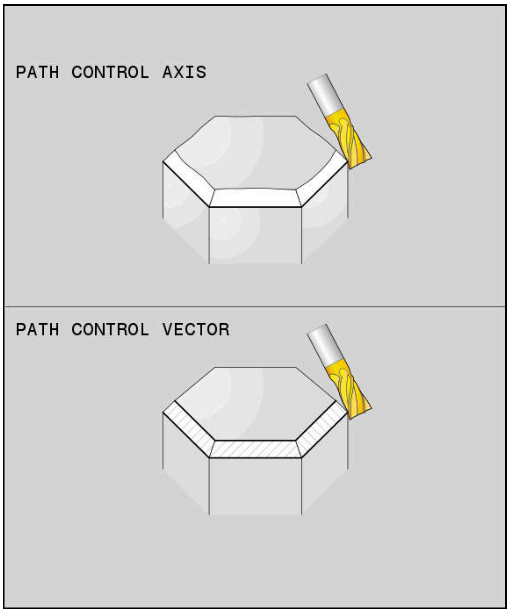
...	
13 FUNCTION TCPM F TCP AXIS POS ...	Rotary axis coordinates are axis angles
...	
18 FUNCTION TCPM F TCP AXIS SPAT ...	Rotary axis coordinates are spatial angles
20 L A+0 B+45 C+0 F MAX	Set tool orientation to B+45 degrees (spatial angle). Define space angle A and C with 0
...	

Orientation interpolation between the start position and end position

With these functions, you define how the tool orientation between the programmed start position and end position are to be interpolated:

- **PATHCTRL AXIS** specifies that the rotary axes between the start position and end position are to be linearly interpolated. The surfaces that arise through milling with the tool circumference (**peripheral milling**) are not necessarily level, and they depend on the machine kinematics.

- **PATHCTRL VECTOR** specifies that the tool orientation within the NC block always lies in the plane that is defined through the start orientation and end orientation. If the vector lies between the start position and end position in this plane, then milling with the tool circumference (**peripheral milling**) will produce a level surface.



In both cases, the programmed tool reference point is moved along a straight line between the start position and end position.

> To obtain the most continuous multi-axis movement possible, define Cycle 32 with a **tolerance for rotary axes**.
>
> **Further information:** Cycle Programming User's Manual

PATHCTRL AXIS

You can use the **PATHCTRL AXIS** variant for NC programs with small orientation changes per NC block. In this case, the angle **TA** in Cycle 32 can be large.

You can use **PATHCTRL AXIS** both for face milling and also for peripheral milling.

Further information: "Running CAM programs", Page 461

> HEIDENHAIN recommends the **PATHCTRL AXIS** variant. This enables smooth motion, which has a beneficial effect on the surface quality.

PATHCTRL VECTOR

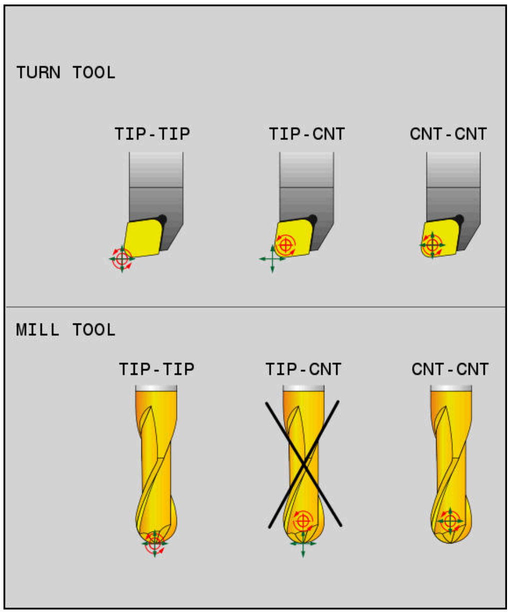
You can use the **PATHCTRL VECTOR** variant for peripheral milling with large orientation changes per NC block.

Example

...	
13 FUNCTION TCPM F TCP AXIS SPAT PATHCTRL AXIS	The rotary axes are linearly interpolated between the start and end positions of the NC block.
14 FUNCTION TCPM F TCP AXIS SPAT PATHCTRL VECTOR	The rotary axes are interpolated such that the tool vector within the NC block always lies in the pane that is specified through the start orientation and end orientation.
...	

Selection of tool reference point and center of rotation

The control provides the following functions for defining the tool reference point and center of rotation:



REF POINT TIP-TIP

- **REFPNT TIP-TIP** references the (theoretical) tool tip for positioning. The center of rotation is also located at the tool tip

REF POINT TIP-CNT

- **REFPNT TIP-CENTER** references the tool tip for positioning. With a milling cutter, the control references the theoretical tool tip for positioning, with a turning tool, it references the virtual tool tip. The center of rotation is located at the center of the cutting-edge radius.

REF POINT CNT-CNT

- **REFPNT CENTER-CENTER** references the center of the cutting-edge radius for positioning. The center of rotation is also located at the center of the cutting-edge radius.

The reference point is optional. If you do not enter anything, the control uses **REFPNT TIP-TIP**.

REFPNT TIP-TIP

The **REFPNT TIP-TIP** variant corresponds to the default behavior of **FUNCTION TCPM**. You can use all previously allowed cycles and functions.

REFPNT TIP-CENTER

The **REFPNT TIP-CENTER** variant is mainly intended for the use with turning tools. In this case the center of rotation and the positioning point are not coincident. In an NC block, the center of rotation (center of the cutting-edge radius) is kept in position, but at the end of the block, the tool tip will no longer be in its initial position.

The main goal of selecting this reference point is to enable machining of complex contours in turning mode with active radius compensation and simultaneously inclined tilting axes (simultaneous turning).

Further information: "Simultaneous turning", Page 527

REFPNT CENTER-CENTER

You can use the **REFPNT CENTER-CENTER** variant to machine parts with a tool whose tip is used as a reference point when executing NC programs generated in a CAD/CAM software where the paths reference the center of the cutting edge radius instead of the tool tip.

Previously, this functionality could only be achieved by shortening the tool with **DL**. The variant with **REFPNT CENTER-CENTER** is advantageous in that the control knows the true tool length and can protect it with **DCM**.

If you use **REFPNT CENTER-CENTER**, to program pocket milling cycles, the control generates an error message.

Example

...	
13 FUNCTION TCPM F TCP AXIS SPAT PATHCTRL AXIS REFPNT TIP-TIP	Both the tool reference point and the center of rotation are located at the tool tip.
14 FUNCTION TCPM F TCP AXIS POS PATHCTRL AXIS REFPNT CENTER-CENTER	Both the tool reference point and the center of rotation are located at the center of the cutting-edge radius.
...	

Resetting FUNCTION TCPM

RESET TCPM

- **FUNCTION RESET TCPM** is to be used if you want to purposely reset the function within an NC program.

> When you select a new NC program in the **Program run, single block** or **Program run, full sequence** operating modes, the control automatically resets the **TCPM** function.

Example

...	
25 FUNCTION RESET TCPM	Resetting FUNCTION TCPM
...	

11.6 Three-dimensional tool compensation (option 9)

Introduction

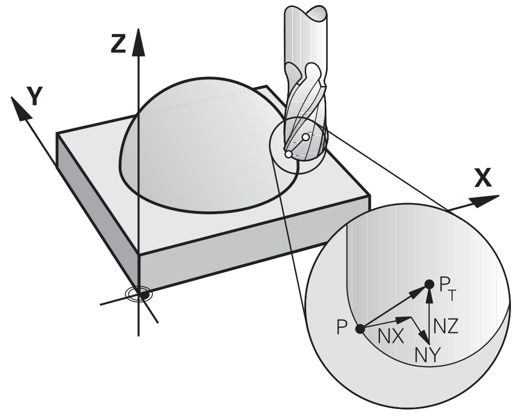
The control can perform a three-dimensional tool compensation (3-D compensation) for straight line blocks. Apart from the X, Y, and Z coordinates of the straight-line end point, these NC blocks must also contain the components NX, NY, and NZ of the surface-normal vector.

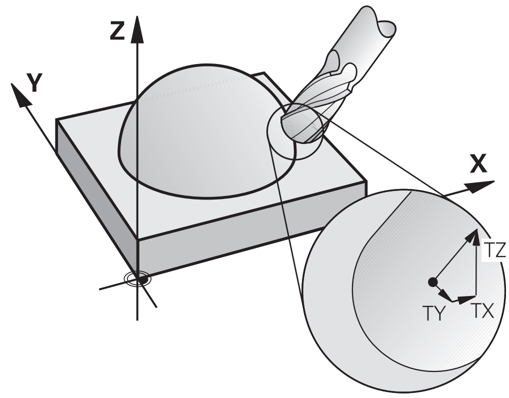
Further information: "Definition of a normalized vector", Page 451

If you want to carry out a tool orientation, these NC blocks need also a normalized vector with the components TX, TY, and TZ, which determines the tool orientation.

Further information: "Definition of a normalized vector", Page 451

The straight-line end point, the components for the surface normals as well as those for the tool orientation must be calculated by a CAM system.





Possible applications

- Use of tools with dimensions that do not correspond with the dimensions calculated by the CAM system (3-D compensation without definition of the tool orientation).
- Face milling: compensation of the cutter geometry in the direction of the surface-normal vector (3-D compensation with and without definition of the tool orientation). Cutting is usually with the end face of the tool.
- Peripheral milling: compensation of the cutter radius perpendicular to the direction of movement and perpendicular to the tool direction (3D radius compensation with definition of the tool orientation). Cutting is usually with the lateral surface of the tool.

Suppressing error messages with positive tool oversize: M107

Standard behavior

With positive tool compensation, programmed contours may be damaged. For NC programs with surface-normal blocks, the control checks whether critical oversizes result from tool compensations, and issues an error message if this is the case.

With Peripheral Milling the control triggers an error message in the following case:

- $DR_{Tab} + DR_{Prog} > 0$

With Face Milling the control triggers an error message in the following case:

- $DR_{Tab} + DR_{Prog} > 0$
- $R2 + DR2_{Tab} + DR2_{Prog} > R + DR_{Tab} + DR_{Prog}$
- $R2 + DR2_{Tab} + DR2_{Prog} < 0$
- $DR2_{Tab} + DR2_{Prog} > 0$

Behavior with M107

With **M107** the control suppresses the error message.

Effect

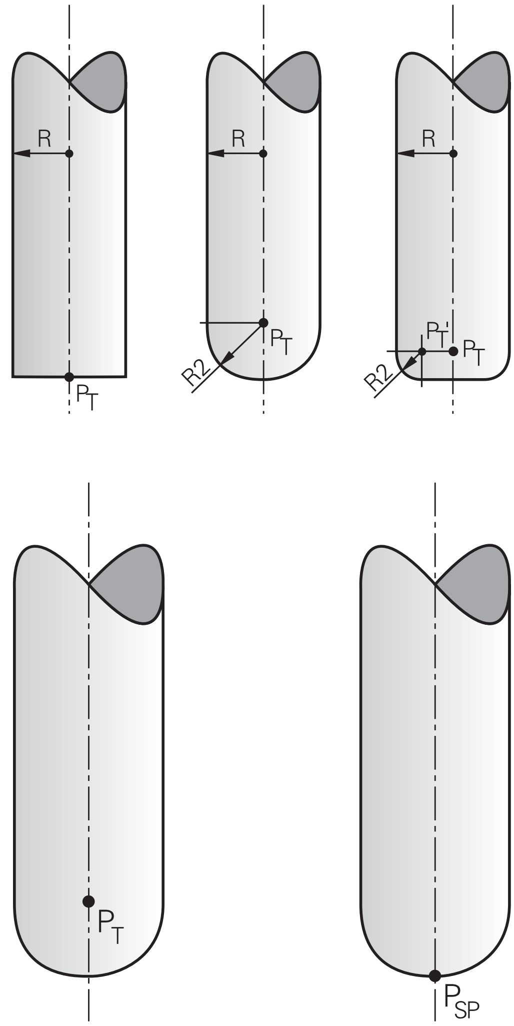
M107 takes effect at the end of block.

You can reset **M107** with **M108**.

> With the **M108** function you can also have the radius of a replacement tool be checked even if three-dimensional tool compensation is not active.

Definition of a normalized vector

A normalized vector is a mathematical quantity possessing a magnitude of 1 and a direction. For LN blocks, the control requires up to two normalized vectors: one in order to determine the direction of the surface normals, and another (optional) to determine the direction of the tool orientation. The direction of a surface normal is determined by the components NX, NY, and NZ. In the case of an end mill and a Ball-nose cutter, the direction of the surface normals points away perpendicularly from the workpiece surface toward the tool reference point PT. A toroid cutter offers the possibilities PT’ or PT (see Figure). The direction of tool orientation is determined by the components TX, TY, and TZ.



Programming notes:

- In the NC syntax, the order must be X,Y, Z for the position and NX, NY, NZ as well as TX, TY, TZ for the vectors.
- The NC syntax of LN blocks must always indicate all of the coordinates and all of the surface-normal vectors, even if the values have not changed from the previous NC block.
- To avoid possible feed interruptions during machining, precisely calculate the vectors and output them (recommended to seven decimal places).
- The 3-D tool compensation using surface normal vectors is effective for the coordinate data specified for the main axes X, Y, Z.
- If you load a tool with oversize (positive delta value), the control generates an error message. You can suppress the error message with the **M107** function.
- The control will not warn you if there is a danger of contour damage due to tool oversizes.

Permissible tool shapes

You can describe the permissible tool shapes in the tool table via tool radii **R** and **R2**:

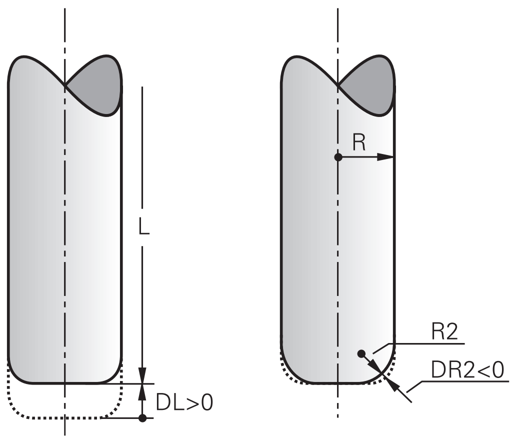
- Tool radius **R**: Distance from the tool center to the tool circumference
- Tool radius 2 **R2**: Radius of the curvature between the tool tip and tool circumference

The value of **R2** generally determines the shape of the tool:

- **R2** = 0: End mill
- **R2** > 0: Toroid cutter (**R2** = **R**: Ball-nose cutter)

These data also provide the coordinates of the tool datum **PT**.

Using other tools: Delta values



If you use tools that have different dimensions from those of the originally programmed tools, then you can enter the difference between the tool lengths and radii as delta values in the tool table or in the NC program:

- Positive delta value **DL, DR**: The tool is larger than the original tool (oversize)
- Negative delta value **DL, DR**: The tool is smaller than the original tool (undersize)

The tool then compensates for the tool position by the sum of the delta values from the tool table and the programmed tool compensation (tool call or compensation table).

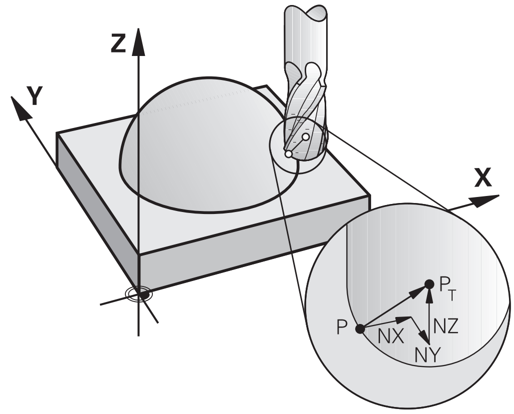
With **DR 2** you modify the rounding radius of the tool and therefore also the tool shape.

If you work with **DR 2** the following applies:

- $R2 + DR2_{Tab} + DR2_{Prog} =$ End mill
- $0 < R2 + DR2_{Tab} + DR2_{Prog} < R$: Toroid cutter
- $R2 + DR2_{Tab} + DR2_{Prog} = R$: Ball-nose cutter

3-D compensation without TCPM

If the NC program includes surface normal vectors, the control performs a 3-D compensation for three-axis machining. In this case, the **RL/RR** radius compensation and **TCPM** or **M128** must be inactive. The control displaces the tool in the direction of the surface-normal vectors by the total of the delta values (from the tool table and **TOOL CALL**).



> The control generally uses the defined **delta values** for 3-D tool compensation. The entire tool radius **R** + **DR**) is only taken into account if you have activated the **FUNCTION PROG PATH IS CONTOUR** function.
>
> **Further information:** "Interpretation of the programmed path", Page 457

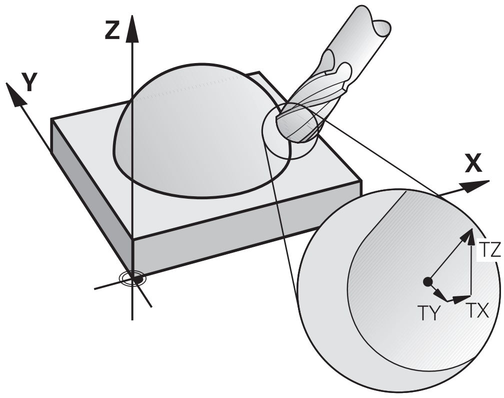
Example: Block format with surface-normal vectors

```
1 LN  X+31.737 Y+21.954  Z+33.165NX+0.2637581 NY+0.0078922
    NZ-0.8764339 F1000  M3
```

LN:	Straight line with 3-D compensation
X, **Y**, **Z**:	Compensated coordinates of the straight-line end point
NX, **NY**, **NZ**:	Components of the surface-normal vector
F:	Feed rate
M:	Miscellaneous function

Face Milling: 3D compensation with TCPM

Face milling is a machining operation carried out with the front face of the tool. If the NC program contains surface-normal vectors and **TCPM** or **M128** is active, 3-D compensation is executed with 5-axis machining. Radius compensation RL/RR must not be active in this case. The control displaces the tool in the direction of the surface-normal vectors by the total of the delta values (from the tool table and **TOOL CALL**).



The control generally uses the defined **delta values** for 3-D tool compensation. The entire tool radius **R** + **DR**) is only taken into account if you have activated the **FUNCTION PROG PATH IS CONTOUR** function.

Further information: "Interpretation of the programmed path", Page 457

If no tool orientation was defined in the **LN** block and **TCPM** is active, the control maintains the tool perpendicular to the workpiece contour.

Further information: "Retain position of the tool tip during the positioning of tilting axes (TCPM): M128 (option 9)", Page 438

If a tool orientation **T** has been defined in the **LN** block and M128 (or **FUNCTION TCPM**) is active at the same time, then the control will position the rotary axes automatically in such a way that the tool can reach the specified tool orientation. If you have not activated **M128** (or **TCPM FUNCTION**), then the control ignores the direction vector **T,** even if it is defined in the **LN** block.

Refer to your machine manual!

The control is not able to automatically position the rotary axes on all machines.

NOTICE

Danger of collision!

The rotary axes of a machine may have limited ranges of traverse, e.g. between -90° and +10° for the B head axis. Changing the tilt angle to a value of more than +10° may result in a 180° rotation of the table axis. There is a danger of collision during the tilting movement!

- Program a safe tool position before the tilting movement, if necessary.
- Carefully test the NC program or program section in the **Program run, single block** operating mode

Example: Block format with surface-normal vectors without tool orientation

```
LN  X+31.737 Y+21.954 Z+33.165 NX+0.2637581 NY+0.0078922 NZ-
    0.8764339 F1000 M128
```

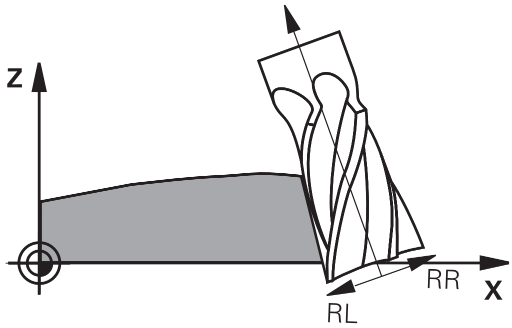
Example: Block format with surface-normal vectors and tool orientation

```
LN  X+31.737 Y+21.954 Z+33.165  NX+0.2637581 NY+0.0078922
    NZ-0.8764339 TX+0.0078922 TY-0.8764339 TZ+0.2590319
    F1000 M128
```

LN:	Straight line with 3-D compensation
X, **Y**, **Z**:	Compensated coordinates of the straight-line end point
NX, **NY**, **NZ**:	Components of the surface-normal vector
TX, **TY**, **TZ**:	Components of the normalized vector for workpiece orientation
F:	Feed rate
M:	Miscellaneous function

Peripheral milling: 3-D radius compensation with TCPM and radius compensation (RL/RR)

The control offsets the tool perpendicularly to the direction of motion and perpendicularly to the direction of the tool by the sum of the delta values **DR** (tool table and NC program). Determine the compensation direction with radius compensation **RL/RR** (see figure, traverse direction Y+). In order for the control to be able to reach the specified tool orientation, you need to activate the function **M128** or **TCPM**.



Further information: "Retain position of the tool tip during the positioning of tilting axes (TCPM): M128 (option 9)", Page 438

The control then positions the rotary axes automatically in such a way that the tool can reach the specified tool orientation with the active compensation.

Refer to your machine manual!

This function exclusively only available with spatial angles. Your machine tool builder defines how these can be entered.

The control is not able to automatically position the rotary axes on all machines.

The control generally uses the defined **delta values** for 3-D tool compensation. The entire tool radius **R** + **DR**) is only taken into account if you have activated the **FUNCTION PROG PATH IS CONTOUR** function.

Further information: "Interpretation of the programmed path", Page 457

NOTICE

Danger of collision!

The rotary axes of a machine may have limited ranges of traverse, e.g. between -90° and +10° for the B head axis. Changing the tilt angle to a value of more than +10° may result in a 180° rotation of the table axis. There is a danger of collision during the tilting movement!

- Program a safe tool position before the tilting movement, if necessary.
- Carefully test the NC program or program section in the **Program run, single block** operating mode

There are two ways to define the tool orientation:

- In an LN block with the components TX, TY and TZ
- In an L block by indicating the coordinates of the rotary axes

Example: Block format with tool orientation

```
1 LN X+31.737 Y+21.954 Z+33.165 TX+0.0078922 TY-0.8764339 TZ
    +0.2590319 RR F1000 M128
```

LN:	Straight line with 3-D compensation
X, **Y**, **Z**:	Compensated coordinates of the straight-line end point
TX, **TY**, **TZ**:	Components of the normalized vector for workpiece orientation
RR:	Tool radius compensation
F:	Feed rate
M:	Miscellaneous function

Example: Block format with rotary axes

```
1 L X+31.737 Y+21.954 Z+33.165 B+12.357 C+5.896 RL F1000
    M128
```

L:	Straight line
X, **Y**, **Z**:	Compensated coordinates of the straight-line end point
B, **C**:	Coordinates of the rotary axes for tool orientation
RL:	Radius Compensation
F:	Feed rate
M:	Miscellaneous function

Interpretation of the programmed path

With the **FUNCTION PROG PATH** function, you decide whether the control will apply the 3-D radius compensation only to the delta values, just as before, or rather to the entire tool radius. If you activate **FUNCTION PROG PATH**, the programmed coordinates exactly correspond to the contour coordinates. With **FUNCTION PROG PATH OFF**, you deactivate this special interpretation.

Procedure

Proceed as follows for the definition:

SPEC FCT
- Show the soft key row with special functions

PROGRAM FUNCTIONS
- Press the **PROGRAM FUNCTIONS** soft key

FUNCTION PROG PATH
- Press the **FUNCTION PROG PATH** soft key

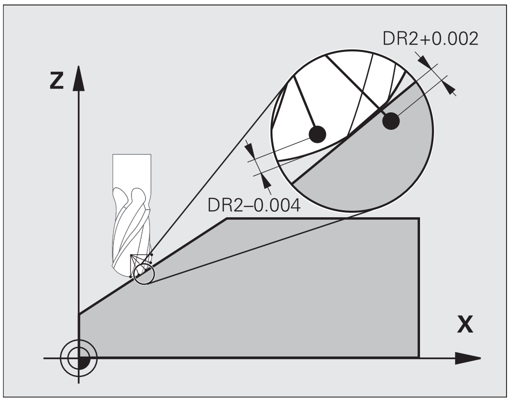
You have the following possibilities:

Soft key	Function
IS CONTOUR	Activate the interpretation of the programmed path as the contour The control takes the full tool radius **R + DR** and the full corner radius **R2 + DR2** into account for 3-D radius compensation.
OFF	Deactivate the special interpretation of the programmed path The control only uses the delta values **DR** and **DR2** for 3-D radius compensation.

If you activate **FUNCTION PROG PATH**, the interpretation of the programmed path as the contour is effective for 3-D compensation movements until you deactivate the function.

3-D radius compensation depending on the tool's contact angle (option 92)

Application



Due to the production process, the effective spherical radius of a ball cutter deviates from the ideal form. The maximum form inaccuracy is defined by the tool manufacturer. Common deviations lie between 0.005 mm and 0.01 mm.

The form inaccuracy can be saved in the form of an compensation value table. This table contains angle values and the deviation from the nominal radius **R2** measured on the respective angle value.

The **3D-ToolComp** software option (option 92) enables the control to compensate the value defined in the compensation value table depending on the actual contact point of the tool.

3-D calibration of the touch probe can also be carried out with the **3D-ToolComp** software option. During this process the deviations determined during touch probe calibration are saved to the compensation value table.

Further information: User's Manual for Setup, Testing and Running NC Programs

Requirements

To be able to use the software option **3D-ToolComp** (option 92) the control requires the following preconditions:

- Option 9 is enabled
- Option 92 is enabled
- The **DR2TABLE** column in the TOOL.T tool table is enabled
- The name of the compensation value table (without its extension) is entered in the **DR2TABLE** column for the tool to be compensated
- 0 is entered in the **DR2** column
- NC program with surface normal vectors (LN blocks)

Error compensation table

If you create the compensation value table yourself, proceed as follows:

- In the file manager open the path **TNC:\system \3D-ToolComp**

- Press the **NEW FILE** soft key
- Enter the file name with extension .3DTC
- > The control opens a table containing the required columns for a compensation value table.

The compensation value table contains three columns:

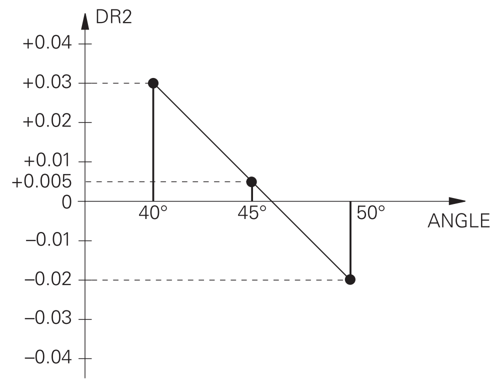
- **NR**: Consecutive line number
- **ANGLE**: Measured angle in degrees
- **DR2**: Radius deviation from the nominal value

The control evaluates a maximum of 100 lines in the compensation value table.

Function

If you are executing an NC program with surface-normal vectors and have assigned a compensation value table (DR2TABLE column) to the active tool in the tool table (TOOL.T), the control uses the values from the compensation value table instead of the compensation value DR2 from TOOL.T.

In doing so, the control takes the compensation value from the compensation value table defined for the current contact point of the tool with workpiece into account. If the contact point is between two compensation points, the control interpolates the compensation value linearly between the two closest angles.



Angle value	Compensation value
40°	0.03 mm (measured)
50°	-0.02 mm (measured)
45° (contact point)	+0.005 mm (interpolated)

Operating and programming notes:

- If the control cannot interpolate a compensation value, it displays an error message.
- **M107** (suppress error message for positive compensation values) is not required, even if positive compensation values are determined.
- The control uses either DR2 from TOOL.T or a compensation value from the compensation value table. Additional offsets, such as a surface oversize, can be defined via DR2 in the NC program (compensation table **.tco** or **TOOL CALL** block).

NC program

The software option **3D-ToolComp** (option 92) only functions with NC programs containing surface normal vectors.

Pay attention when creating the CAM program how you measure the tools:

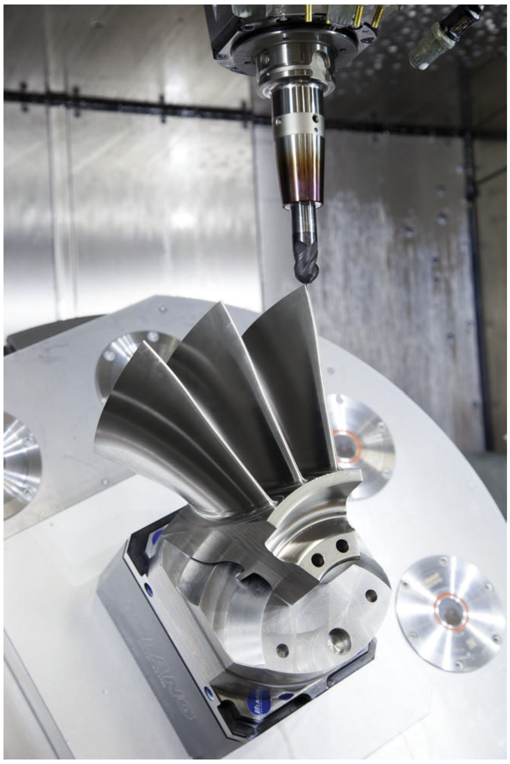
- NC program output at the south pole of the sphere requires tools measured on the tool tip
- NC program output at the center of the sphere requires tools measured on the tool center

11.7 Running CAM programs

If you create NC programs externally using a CAM system, you should pay attention to the recommendations detailed below. This will enable you to optimally use the powerful motion control functionality provided by the control and usually create better workpiece surfaces with shorter machining times. Despite high machining speeds, the control still achieves a very high contour accuracy. The basis for this is the HEROS 5 real-time operating system in conjunction with the **ADP** (Advanced Dynamic Prediction) function of the TNC 640. This enables the control to also efficiently process NC programs with high point densities.

From 3-D model to NC program

Here is a simplified description of the process for creating an NC program from a CAD model:



- **CAD: Model creation**
  Construction departments prepare a 3-D model of the workpiece to be machined. Ideally the 3-D model is designed for the center of tolerance.
- **CAM: Path generation, tool compensation**
  The CAM programmer specifies the machining strategies for the areas of the workpiece to be machined. The CAM system uses the surfaces of the CAD model to calculate the paths of the tool movements. These tool paths consist of individual points calculated by the CAM system so that each surface to be machined is approximated as nearly as possible while considering chord errors and tolerances. This way, a machine-neutral NC program is created, known as a CLDATA file (cutter location data). A postprocessor generates a machine- and control-specific NC program, which can be processed by the CNC control. The postprocessor is adapted according to the machine tool and the control. The postprocessor is the link between the CAM system and the CNC control.
- **Control: Motion control, tolerance monitoring, velocity profile**
  The control uses the points defined in the NC program to calculate the movements of each machine axis as well as the required velocity profiles. Powerful filter functions then process and smooth the contour so that the control does not exceed the maximum permissible path deviation.
- **Mechatronics: Feed control, drive technology, machine tool**
  The motions and velocity profiles calculated by the control are realized as actual tool movements by the machine's drive system.

Consider with post processor configuration

Take the following points into account with post processor configuration:

- Always set the data output for axis positions to at least four decimal places. This way you improve the quality of the NC data and avoid rounding errors, which can result in defects visible to the naked eye on the workpiece surface. Output of five decimal places may achieve improved surface quality for optical components as well as components with very large radii (i.e. small curvatures), for example molds for the automotive industry
- Always set the data output for the machining of surface normal vectors (LN blocks, only Klartext conversational programming) to exactly seven decimal places
- Avoid using successive incremental NC blocks because this may lead to the tolerances of the individual NC blocks being added together in the output
- Set the tolerance in Cycle 32 so that in standard behavior it is at least twice as large as the chord error defined in the CAM system Also note the information describing the functioning of Cycle 32.
- If the chord error selected in the CAM program is too large, then, depending on the respective curvature of a contour, large distances between NC blocks can result, each with large changes of direction. During machining this leads to drops in the feed rate at the block transitions. Recurring and equal accelerations (i.e. force excitation), caused by feed-rate drops in the heterogeneous NC program, can lead to undesirable excitation of vibrations in the machine structure.
- You can also use arc blocks instead of linear blocks to connect the path points calculated by the CAM system. The control internally calculates circles more accurately than can be defined via the input format
- Do not output any intermediate points on exactly straight lines. Intermediate points that are not exactly on a straight line can result in defects visible to the naked eye on the workpiece surface
- There should be exactly one NC data point at curvature transitions (corners)
- Avoid sequences of many short block paths. Short paths between blocks are generated in the CAM system when there are large curvature transitions with very small chord errors in effect. Exactly straight lines do not require such short block paths, which are often forced by the continuous output of points from the CAM system
- Avoid a perfectly even distribution of points over surfaces with a uniform curvature, since this could result in patterns on the workpiece surface
- For 5-axis simultaneous programs: avoid the duplicated output of positions if they only differ in the tool’s angle of inclination
- Avoid the output of the feed rate in every NC block. This would negatively influence the control’s velocity profile

Useful configurations for the machine tool operator:

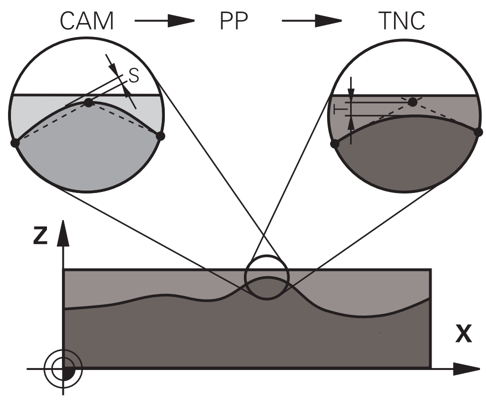
- In order to improve the structure of large NC programs, use the control's structuring function
  Further information: "Structuring NC programs", Page 196
- Use the control's commenting function in order to document NC programs
  Further information: "Adding comments", Page 192
- When machining holes and simple pocket geometries, use the comprehensive cycles available in the control
  Further information: Cycle Programming User's Manual
- For fits, output the contours with **RL/RR** tool radius compensation. This makes it easy for the machine operator to make necessary compensations
  Further information: "Tool compensation", Page 133
- Separate feed rates for pre-positioning, machining, and downfeeds, and define them via Q parameters at the beginning of the program

Example: Variable feed rate definitions

1 Q50 = 7500	FEED RATE FOR POSITIONING
2 Q51 = 750	FEED RATE FOR PLUNGING
3 Q52 = 1350	FEED RATE FOR MILLING
...	
25 L Z+250 R0 FMAX	
26 L X+235 Y-25 FQ50	
27 L Z+35	
28 L Z+33.2571 FQ51	
29 L X+321.7562 Y-24.9573 Z+33.3978 FQ52	
30 L X+320.8251 Y-24.4338 Z+33.8311	
...	

Please note the following for CAM programming

Adapting chord errors



Programming notes:

- For finishing operations, do not set the chord error in the CAM system to a value greater than 5 µm. In Cycle 32, use an appropriate tolerance factor **T** of 1.3 to 3.
- For roughing operations, the total of the chord error and the tolerance **T** must be less than the defined machining oversize. In this way you can avoid contour damage.
- The specific values depend upon the dynamics of your machine.

Adapt the chord error in the CAM program, depending on the machining:

- **Roughing with preference for speed:**
  Use higher values for the chord error and the matching tolerance value in Cycle 32. Both values depend on the oversize required on the contour. If a special cycle is available on your machine, use the roughing mode. In roughing mode the machine generally moves with high jerk values and high accelerations
  - Normal tolerance in Cycle 32: Between 0.05 mm and 0.3 mm
  - Normal chord error in the CAM system: Between 0.004 mm and 0.030 mm
- **Finishing with preference for high accuracy:**
  Use smaller values for the chord error and a matching low tolerance in Cycle 32. The data density must be high enough for the control to detect transitions and corners exactly. If a special cycle is available on your machine, use the finishing mode.
  In finishing mode the machine generally moves with low jerk values and low accelerations
  - Normal tolerance in Cycle 32: Between 0.002 mm and 0.006 mm
  - Normal chord error in the CAM system: Between 0.001 mm and 0.004 mm
- **Finishing with preference for high surface quality:**
  Use small values for the chord error and a matching larger tolerance in Cycle 32. The control is then able to better smooth the contour. If a special cycle is available on your machine, use the finishing mode. In finishing mode the machine generally moves with low jerk values and low accelerations
  - Normal tolerance in Cycle 32: Between 0.010 mm and 0.020 mm
  - Normal chord error in the CAM system: Approx. 0.005 mm

Further adaptations

Take the following points into account with CAM programming:

- For slow machining feed rates or contours with large radii, define the chord error to be only one-third to one-fifth of tolerance **T** in Cycle 32. Additionally, define the maximum permissible point spacing to be between 0.25 mm and 0.5 mm. The geometry error or model error should also be specified to be very small (max. 1 µm).
- Even at higher machining feed rates, point spacings of greater than 2.5 mm are not recommended for curved contour areas
- For straight contour elements, one NC point at the beginning of a line and one NC point at the end suffice. Avoid the output of intermediate positions
- In programs with five axes moving simultaneously, avoid large changes in the ratio of path lengths in linear and rotational blocks. Otherwise large reductions in the feed rate could result at the tool reference point (TCP)
- The feed-rate limitation for compensating movements (e.g. via **M128 F...**) should be used only in exceptional cases. The feed-rate limitation for compensating movements can cause large reductions in the feed rate at the tool reference point (TCP).
- NC programs for 5-axis simultaneous machining with spherical cutters should preferably be output for the center of the sphere. The NC data are then generally more consistent. Additionally, in Cycle 32 you can set a higher rotational axis tolerance **TA** (e.g. between 1° and 3°) for an even more constant feed-rate curve at the tool reference point (TCP).
- For NC programs for 5-axis simultaneous machining with toroid cutters or spherical cutters, where the NC output is for the south pole of the sphere, choose a lower rotary axis tolerance. 0.1° is a typical value. However, the maximum permissible contour damage is the decisive factor for the rotational axis tolerance. This contour damage depends on the possible tool tilting, tool radius and contact depth of the tool.
  With 5-axis gear hobbing with an end mill you can calculate the maximum possible contour damage T directly from the cutter contact length L and permissible contour tolerance TA:
  T ~ K x L x TA K = 0.0175 [1/°]
  Example: L = 10 mm, TA = 0.1°: T = 0.0175 mm

Possibilities for intervention on the control

Cycle 32 **TOLERANCE** is available for influencing the behavior of CAM programs directly on the control. Please note the information describing the functioning of Cycle 32. Also note the interactions with the chord error defined in the CAM system.

Further information: Cycle Programming User's Manual

> Refer to your machine manual!
>
> Some machine tool builders provide an additional cycle for adapting the behavior of the machine to the respective machining operation, such as Cycle 332 Tuning. Cycle 332 can be used to modify filter settings, acceleration settings, and jerk settings.

Example

```
34 CYCL DEF 32.0 TOLERANCE
35 CYCL DEF 32.1 T0.05
36 CYCL DEF 32.2 HSC MODE:1 TA3
```

ADP motion control

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

An insufficient quality of data in NC programs created on CAM systems frequently causes inferior surface quality of the milled workpieces. The **ADP** (Advanced Dynamic Prediction) feature expands the conventional look-ahead of the permissible maximum feed rate profile and optimizes the motion control of the feed axes during milling. This enables clean surfaces with short machining times to be cut, even with a strongly fluctuating distribution of points in adjacent tool paths. This significantly reduces or eliminates the reworking complexity.

These are the most important benefits of ADP:

- Symmetrical feed-rate behavior on forward and backward paths with bidirectional milling
- Uniform feed rate curves with adjacent cutter paths
- Improved reaction to negative effects (e.g. short, step-like stages, coarse chord tolerances, heavily rounded block end-point coordinates) in NC programs generated by CAM system
- Precise compliance to dynamic characteristics even in difficult conditions

12

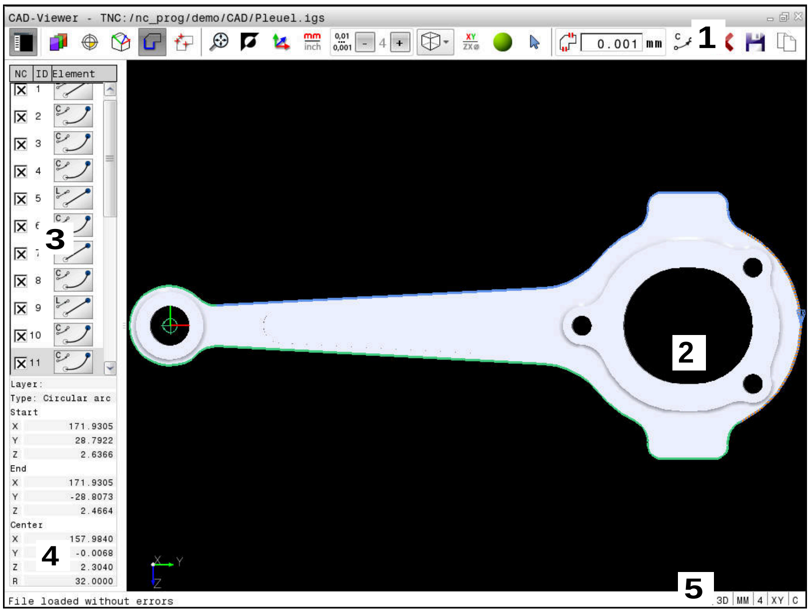
Data Transfer from CAD Files

12.1 Screen layout of the CAD viewer

Fundamentals of the CAD viewer

Screen display

When you open the **CAD-Viewer**, the following screen layout is displayed:



1 Menu bar
2 Graphics window
3 List View window
4 Element Information window
5 Status bar

File types

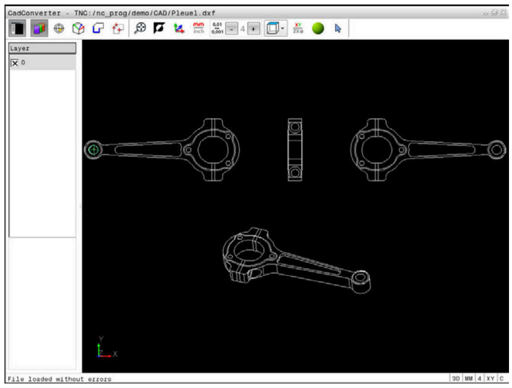
The **CAD-Viewer** enables you to open standardized CAD data formats directly on the control.
The control displays the following file types:

File	Type	Format
Step	.STP and .STEP	■ AP 203 ■ AP 214
IGES	.IGS and .IGES	■ Version 5.3
DXF	.DXF	■ R10 to 2015

12.2 CAD Import (option 42)

Application

You can open CAD files directly on the control in order to extract contours and machining positions from it. You can then store them as Klartext programs or as point files. Klartext programs acquired in this manner can also be run on older HEIDENHAIN controls, since these contour programs contain only **L** and **CC**/**C** blocks.



If you process files in **Programming** mode, then the control generates contour programs with the file extension **.H** and point files with the extension **.PNT** by default. You can select the file type in the save dialog. To insert a selected contour or a selected machining position directly into an NC program, use the control's clipboard.

> Operating notes:
> - Before loading the file into the control, ensure that the name of the file contains only permitted characters. **Further information:** "File names", Page 104
> - The control does not support binary DXF format. Save the DXF file in ASCII format in the CAD or drawing program.

Using the CAD viewer

You need a mouse or touchpad in order to use the **CAD-Viewer** without a touchscreen. All operating modes and functions as well as contours and machining positions can only be selected with the mouse or touch pad.

The **CAD-Viewer** runs as a separate application on the third desktop of the control. This enables you to use the screen switchover key to switch between the machine operating modes, the programming modes and the **CAD-Viewer**. This is particularly useful if you want to add contours or machining positions to a Klartext program by copy and paste using the clipboard.

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

Further information: "Operating the touchscreen", Page 545

Opening the CAD file

- Press the **Programming** key

PGM MGT
- To call the file manager, press the **PGM MGT** key

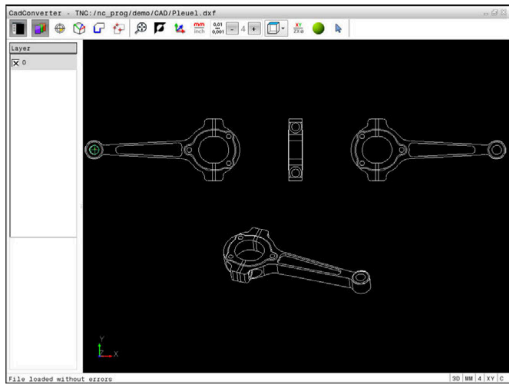
SELECT TYPE
- Select the soft key menu for selecting the file types to be displayed: Press the **SELECT TYPE** soft key

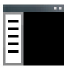
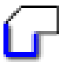
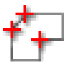
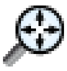
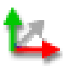
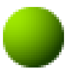
SHOW CAD
- To show all CAD files, press the **SHOW CAD** or **SHOW ALL** soft key
- Select the directory in which the CAD file is saved
- Select the desired CAD file

ENT
- Press the **ENT** key
- > The control starts the **CAD-Viewer** and shows the file contents on the screen. The control displays the layers in the List View window and the drawing in the Graphics window.

Basic settings

The basic settings specified below are selected using the icons in the header bar.



Icon	Setting
	Show or hide the List View window in order to expand the Graphics window
	Display of the various layers
	Set preset, with optional selection of the plane
	Set datum, with optional selection of the plane
	Select the contour
	Select hole positions
	Set the zoom to the largest possible rendering of the entire graphical representation
	Switch background color (black or white)
	Switch between 2-D and 3-D mode. The active mode is highlighted in color
mm inch	Set the unit of measure (**mm** or **inch**) for the file. The control then outputs the contour program and the machining positions in this unit of measure. The active unit of measure is highlighted in red
0,01 ... 0,001	Set resolution: The resolution specifies how many decimal places the control will use when generating the contour program. Default setting: 4 decimal places with **mm** as the unit of measure, and 5 decimal places with **inch** as the unit of measure
	Switch between various views of the model (e.g., **Top**)
XY ZXØ	Select a contour for a turning operation. The active machining operation is highlighted in color (option 50)
	Activate wire model of a 3-D drawing

Icon	Setting
	Selection and deselection: The active **+** symbol is the same as the pressed **Shift** key, and the active **-** symbol is the same as the pressed **CTRL** key. The active **cursor** symbol is the same as the mouse

The control displays the following icons only in a certain modes.

Icon	Setting
	The most recent step is undone.
	Contour transfer mode: The tolerance specifies how far apart neighboring contour elements may be from each other. You can use the tolerance to compensate for inaccuracies that occurred when the drawing was made. The default setting is 0.001 mm
C CR	Arc mode: Arc mode defines whether circular arcs are output in C format or CR format (e.g., for cylinder surface interpolation) in the NC program.
	Point transfer mode: Specifies whether the control should display the tool path as a dashed line during the selection of machining positions
	Path optimization mode: The control optimizes the tool traverse movement such that there are shorter traverse movements between the machining positions. When the icon is pressed repeatedly, the optimization is reset
	Hole position mode: The control opens a pop-up window in which you can filter holes (full circles) based on their size

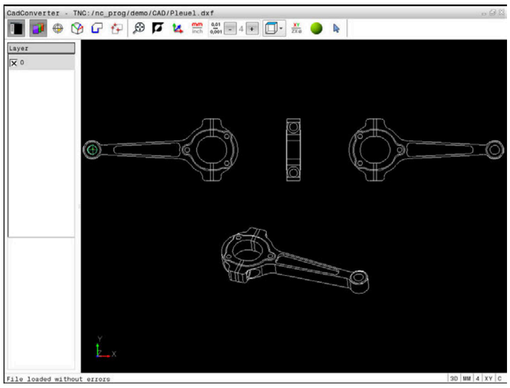
Operating notes:

- Set the correct unit of measure, because the CAD file does not contain this information.
- When creating NC programs for earlier control models, you must limit the resolution to three decimal places. In addition, you must remove the comments that the **CAD-Viewer** outputs into the contour program.
- The control displays the active basic settings in the status bar of the screen.

Setting layers

CAD files usually contain several layers. The designer uses these layers to create groups of various types of elements, e.g. the actual workpiece contour, dimensions, auxiliary and design lines, shadings, and texts.

Hiding unneeded layers makes the graphics easier to read and facilitates the extraction of the required information.

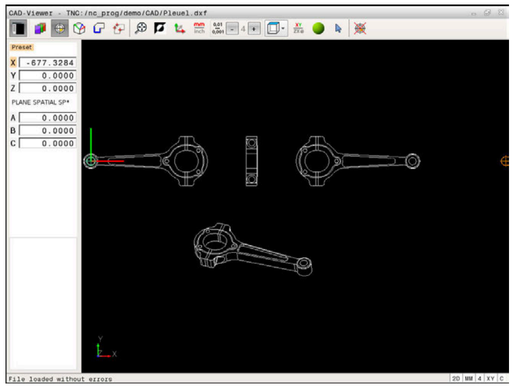


Operating notes:

- The CAD file to be processed must contain at least one layer. Elements not assigned to a layer are automatically moved by the control to the anonymous layer.
- You can even select a contour if the designer has saved the lines on different layers.

- ▶ Select the mode for the layer settings
- > In the List View window the control shows all layers contained in the active CAD file
- ▶ Hide a layer: Select the layer with the left mouse button, and click its check box to hide it
- ▶ Alternatively, use the space key
- ▶ Show a layer: Select the layer with the left mouse button, and click its check box to show it
- ▶ Alternatively, use the space key

Defining a preset



The datum of the drawing in the CAD file is not always located in a manner that lets you use it directly as a workpiece preset. Therefore, the control has a function with which you can shift the workpiece preset to a suitable location by clicking an element. You can also define the orientation of the coordinate system.

You can define a preset at the following locations:

- By directly inputting numerical values into the List View window
- At the beginning, end or center of a straight line
- At the beginning, center or end of a circular arc
- At the transition between quadrants or at the center of a complete circle
- At the intersection between:
  - A straight line and a straight line, even if the intersection is actually on the extension of one of the lines
  - Straight line – circular arc
  - Straight line – full circle
  - Circle – circle (regardless of whether a circular arc or a full circle)

> Operating notes:
> - You can change the preset even after you have selected the contour. The control does not calculate the actual contour data until you save the selected contour in a contour program.

NC syntax

The preset and optional orientation are inserted in the NC program as a comment starting with **origin**.

```
4 ;orgin = X... Y... Z...
5 ;orgin_plane_spatial = SPA... SPB... SPC...
```

Selecting a preset on a single element

- ▶ Select the mode for specifying the preset
- ▶ Click the desired element with the mouse
- > The control indicates possible locations for presets on the selected element with stars.
- ▶ Click the star you want to select as preset
- ▶ Use the zoom function if the selected element is too small
- > The control sets the preset symbol at the selected location.
- > You can align the coordinate system as needed. **Further information:** "Adjusting the orientation of the coordinate system", Page 475

Selecting a preset on the intersection of two elements

- Select the mode for specifying the preset
- Click the first element (straight line, circle or circular arc) with the left mouse button
- > The element is color-highlighted.
- Click the second element (straight line, circle or circular arc) with the left mouse button
- > The control sets the preset symbol on the intersection.
- > You can align the coordinate system as needed. **Further information:** "Adjusting the orientation of the coordinate system", Page 475

Operating notes:
- If there are several possible intersections, the control selects the intersection nearest the mouse-click on the second element.
- If two elements do not intersect directly, the control automatically calculates the intersection of their extensions.
- If the control cannot calculate an intersection, it deselects the previously selected element.

If a preset is set, the color of the "Setting a preset" icon changes.

You can delete a preset by pressing the icon.

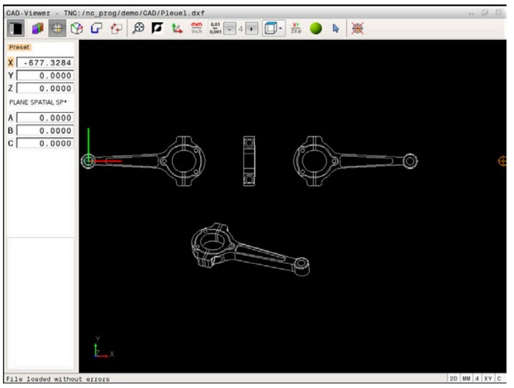
Adjusting the orientation of the coordinate system

The position of the coordinate system is defined by the orientation of the axes.

- The preset has already been set
- Left-click an element that is in the positive X direction
- > The control aligns the X axis and changes the angle in C.
- > The control colors the list view orange if the defined angle does not equal 0.
- Left-click an element that is approximately in the positive Y direction
- > The control aligns the Y and Z axes and changes the angle in A and C.
- > The control colors the list view orange if the defined value does not equal 0.

Element Information

In the Element Information window, the control shows how far the preset you have chosen is located from the drawing datum, and how this reference system is oriented with respect to the drawing.

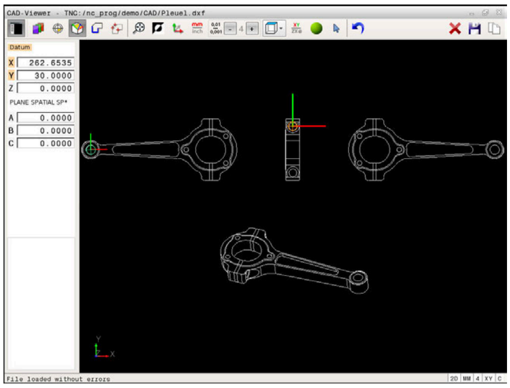


Defining the datum

The workpiece preset is not always located in a manner that lets you machine the entire part. Therefore, the control has a function with which you can define a new datum and a tilting operation.

The datum with the orientation of the coordinate system can be defined at the same positions as a preset.

Further information: "Defining a preset", Page 474



NC syntax

The datum and its optional orientation can be inserted as NC block or comments in the NC program by using the **TRANS DATUM AXIS** function for the datum and the **PLANE SPATIAL** function for the orientation.

If you specify only one datum and its orientation, then the control inserts the functions in the NC program as an NC block.

```
4 TRANS DATUM AXIS X... Y... Z...
5 PLANE SPATIAL SPA... SPB... SPC... TURN MB MAX FMAX
```

If you additionally select contours or points, then the control inserts the functions in the NC program as comments.

```
4 ;TRANS DATUM AXIS X... Y... Z...
5 ;PLANE SPATIAL SPA... SPB... SPC... TURN MB MAX FMAX
```

Selecting the datum on a single element

- ▶ Select the mode for specifying the datum
- ▶ Click the desired element with the mouse
- > The control indicates possible locations for the datum on the selected element with stars.
- ▶ Click the star you want to select as datum
- ▶ Use the zoom function if the selected element is too small
- > The control sets the preset symbol at the selected location.
- > You can align the coordinate system as needed. **Further information:** "Adjusting the orientation of the coordinate system", Page 478

Selecting a datum on the intersection of two elements

- ▶ Select the mode for specifying the datum
- ▶ Click the first element (straight line, circle or circular arc) with the left mouse button
- > The element is color-highlighted.
- ▶ Click the second element (straight line, circle or circular arc) with the left mouse button
- > The control sets the preset symbol on the intersection.
- > You can align the coordinate system as needed.
  Further information: "Adjusting the orientation of the coordinate system", Page 478

> Operating notes:
> - If there are several possible intersections, the control selects the intersection nearest the mouse-click on the second element.
> - If two elements do not intersect directly, the control automatically calculates the intersection of their extensions.
> - If the control cannot calculate an intersection, it deselects the previously selected element.

When a datum has been set, the color of the datum setting icon changes.

You can delete a datum by pressing the icon.

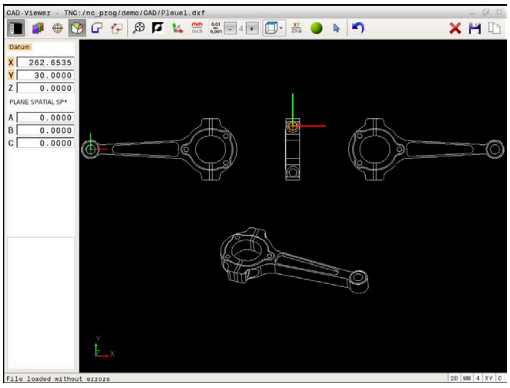
Adjusting the orientation of the coordinate system

The position of the coordinate system is defined by the orientation of the axes.

- ▶ The datum has already been set
- ▶ Left-click an element that is in the positive X direction
- > The control aligns the X axis and changes the angle in C.
- > The control colors the list view orange if the defined angle does not equal 0.
- ▶ Left-click an element that is approximately in the positive Y direction
- > The control aligns the Y axis and Z axis, and changes the angle in A and C.
- > The control colors the list view orange if the defined value does not equal 0.

Element information

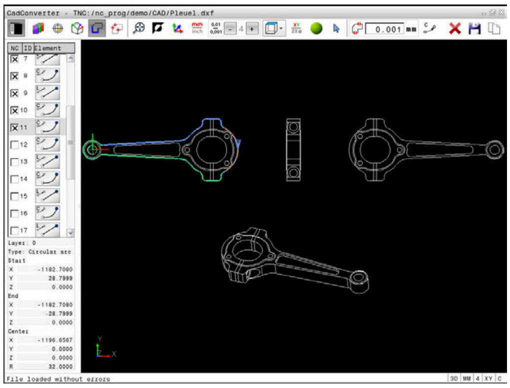
In the Element Information window, the control displays how far away the datum you selected is from the workpiece preset.



Selecting and saving a contour

Operating notes:

- This function is not available if option 42 is not enabled.
- Specify the direction of rotation during contour selection so that it matches the desired machining direction.
- Select the first contour element such that approach without collision is possible.
- If the contour elements are very close to one another, use the zoom function.



The following elements are selectable as contours:

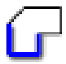
- Line segment
- Circle
- Circular arc
- Polyline

On curved elements, such as splines or ellipses, you can select the end points and center points. They can also be selected as part of contours and converted to polylines during export.

Element information

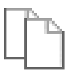
In the Element Information window the control displays a range of information about the last contour element you selected in the List View window or in the Graphics window.

- **Layer**: Indicates the layer you are currently on
- **Type**: Indicates the current element type, e.g. line
- **Coordinates**: Shows the starting point and end point of an element, and circle center and radius where appropriate

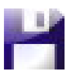


- Select the contour selection mode
- > The Graphics window is active for the contour selection.
- To select a contour element, click the element with the mouse
- > The control displays the machining sequence as a dashed line.
- Position the mouse on the other side of the center point of an element to modify the machining sequence
- Select the element with the left mouse button
- > The selected contour element turns blue.
- > If further contour elements in the selected machining sequence are selectable, the control highlights these elements in green. At junctions, the control chooses the element with the least deviation in direction.
- Click the last green element to add all elements to the contour program
- > The control shows all selected contour elements in the List View window. Elements that are still green are displayed without a check mark in the **NC** column. The control does not save these elements to the contour program.
- You can also add selected elements to the contour program by clicking them in the List View window
- If necessary you can also deselect elements that you already selected by clicking the element in the Graphics window again, but this time while pressing the **CTRL** key

- Alternative: Click the icon to deselect all selected elements



- Save the selected contour elements to the clipboard of the control so that you can then insert the contour in a Klartext program



- Alternative: Save the selected contour elements as a Klartext program
- > The control displays a pop-up window in which you can select the target directory, a file name, and the file type.

ENT

- Confirm the entry
- > The control saves the contour program to the selected directory.

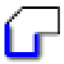
- If you want to select more contours, press the Cancel Selected Elements soft key and select the next contour as described above

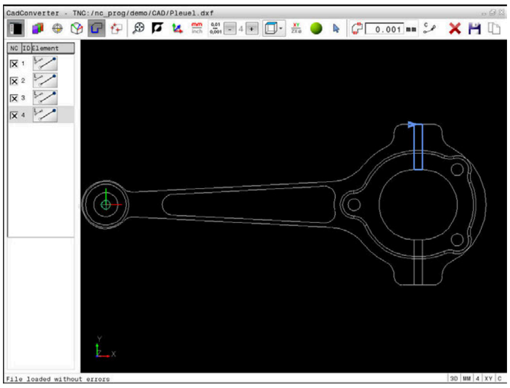
Operating notes:

- The control also transfers two workpiece-blank definitions (**BLK FORM**) to the contour program. The first definition contains the dimensions of the entire CAD file. The second one, which is the active one, contains only the selected contour elements, so that an optimized size of the workpiece blank results.
- The control only saves elements that have been selected (blue elements), which means that they have been given a check mark in the List View window.

Dividing, extending and shortening contour elements

Proceed as follows to modify contour elements:





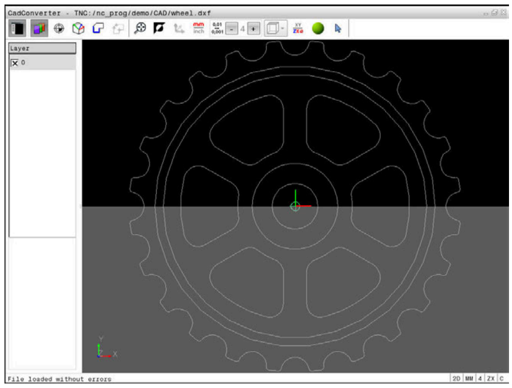
- ▶ The Graphics window is active for the contour selection
- ▶ To select the starting point, select an element or the intersection between two elements (using the **+** icon)
- ▶ Select the next contour element by clicking it with the mouse
- > The control displays the machining sequence as a dashed line.
- ▶ When the element is selected the control displays it in blue.
- > If the elements cannot be connected the control displays the selected element in gray.
- > If further contour elements in the selected machining sequence are selectable, the control highlights these elements in green. At junctions, the control chooses the element with the least deviation in direction.
- ▶ Click the last green element to add all elements to the contour program.

Operating notes:

- You select the machining sequence of the contour with the first contour element.
- If the contour element to be extended or shortened is a straight line, then the control extends or shortens the contour element along the same line. If the contour element to be extended or shortened is a circular arc, then the control extends or shortens the contour element along the same arc.

Selecting a contour for a turning operation

You can also use the CAD viewer (option 50) to select contours for turning. The icon is grayed out if option 50 is not enabled. Before selecting a turning contour, you must set the preset on the rotary axis. If you select a turning contour, it is saved with Z and X coordinates. In addition, all X coordinate values in turning contours are transferred as diameter values, i.e. the drawing dimensions for the X axis are doubled. All contour elements below the rotary axis cannot be selected and are highlighted gray.



- ▶ Select the mode for choosing a turning contour
- > The control shows only the selectable elements above the rotation center.
- ▶ Select the desired contour elements with the left mouse button
- > The control displays the selected contour elements in blue and shows the selected elements with a symbol (circular or straight) in the List View window.

> The icons specified above have identical functions for both milling and turning. Icons not available for turning are disabled.

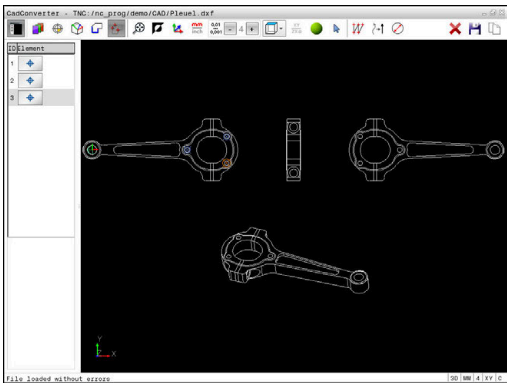
You can also use the mouse to change the turning graphic display. The following functions are available:

- ▶ To shift the model shown: Hold the center mouse button or the wheel button down and move the mouse.
- ▶ To zoom in on a certain area: Mark a zoom area by holding the left mouse button down. After you release the left mouse button, the control zooms in on the defined area
- ▶ To rapidly magnify or reduce any area: Rotate the mouse wheel backwards or forwards
- ▶ To return to the standard display: Double-click with the right mouse key

Selecting and saving machining positions

Operating notes:

- This function is not available if option 42 is not enabled.
- If the contour elements are very close to one another, use the zoom function.
- If required, configure the basic settings so that the control shows the tool paths. **Further information:** "Basic settings", Page 471



Three possibilities are available in the pattern generator for defining machining positions:

- Single selection: You select the desired machining position through individual mouse clicks
  Further information: "Single selection", Page 485
- Rapid selection of hole positions with the mouse area: By dragging the mouse to define an area, you can select all the hole positions within this area
  Further information: "Rapid selection of hole positions via mouse area", Page 486
- Rapid selection of hole positions via an icon: Click the icon and the control then displays all existing hole diameters
  Further information: "Rapid selection of hole positions via icon", Page 487

Selecting the file type

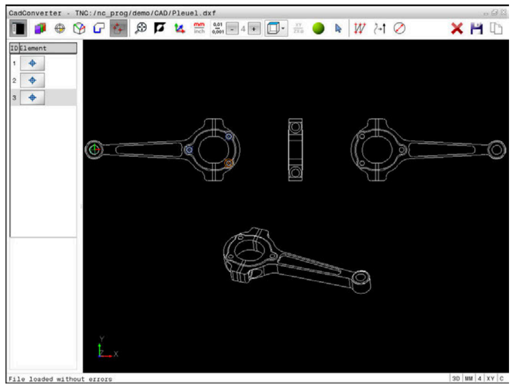
The following file types are available:

- Point table (**.PNT**)
- Klartext conversational language program (**.H**)

If you save the machining positions to a Klartext program, the control creates a separate linear block with cycle call for every machining position (**L X... Y... Z... F MAX M99**). You can also transfer this NC program to older HEIDENHAIN controls and run it there.

The point tables (.PNT) of the TNC 640 and iTNC 530 are not compatible. Transferring and processing on the other control type in each case may lead to problems and unforeseen performance.

Single selection



- Select the mode for choosing a machining position
- The Graphics window is active for position selection.
- To select a machining position, click the element with the mouse
- The control displays the element in orange.
- If the shift key is pressed at the same time, the control indicates possible machining positions on the element with stars.
- If you click a circle, the control adopts the circle center as machining position
- If the shift key is pressed at the same time, the control indicates possible machining positions with stars.
- The control loads the selected position into the List View window (displays a point symbol).
- If necessary you can also deselect elements that you already selected by clicking the element in the Graphics window again, but this time while pressing the CTRL key
- Alternative: Select the element in the List View window and press the **DEL** key
- Alternative: Click the icon to deselect all selected elements
- Save the selected machining positions to the clipboard of the control so that you can then insert them as a positioning block with cycle call in a Klartext program
- Alternative: Save the selected machining positions in a point file
- The control displays a pop-up window in which you can select the target directory, a file name, and the file type.

ENT

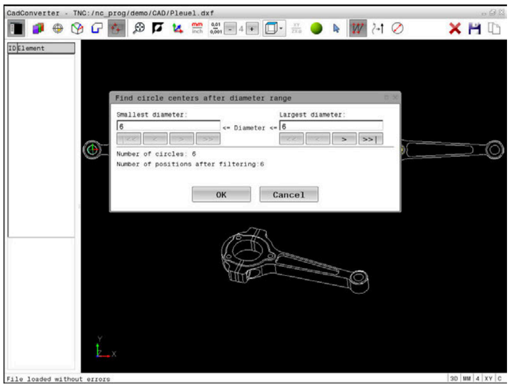
- Confirm the entry
- The control saves the contour program to the selected directory.
- If you want to select more machining positions, press the Cancel Selected Elements icon and select as described above

Rapid selection of hole positions via mouse area

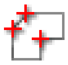
- Select the mode for choosing a machining position
- > The Graphics window is active for position selection.
- To select machining positions, press the shift key and define an area with the left mouse button
- > All complete circles that are fully enclosed within the area are adopted as hole positions by the control.
- > The control opens a pop-up window in which you can filter the holes by size.
- Configure the filter settings and press the **OK** button to confirm
  Further information: "Filter settings", Page 488
- > The control loads the selected positions into the List View window (displays a point symbol).
- If necessary you can also deselect elements that you already selected by clicking the element in the Graphics window again, but this time while pressing the CTRL key
- Alternative: Select the element in the List View window and press the **DEL** key
- Alternative: Deselect all elements by dragging an area open again, but this time while pressing the CTRL key
- Save the selected machining positions to the clipboard of the control so that you can then insert them as a positioning block with cycle call in a Klartext program
- Alternative: Save the selected machining positions in a point file
- > The control displays a pop-up window in which you can select the target directory, a file name, and the file type.

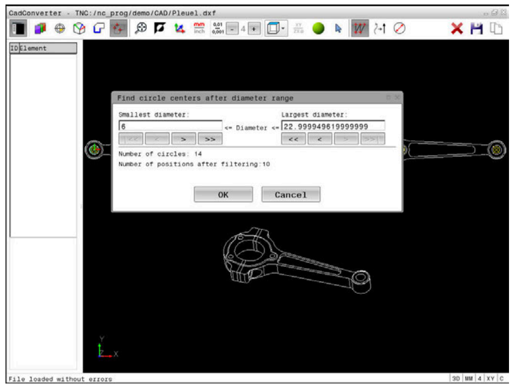
ENT

- Confirm the entry
- > The control saves the contour program to the selected directory.
- If you want to select more machining positions, press the Cancel Selected Elements icon and select as described above

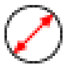


Rapid selection of hole positions via icon



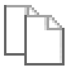


- ▶ Select the mode for choosing machining positions
- > The Graphics window is active for position selection.

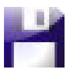


- ▶ Select the icon
- > The control opens a pop-up window in which you can filter bore holes (full circles) by size.
- ▶ Configure the filter settings if required and press the **OK** button to confirm
  Further information: "Filter settings", Page 488
- > The control loads the selected positions into the List View window (displays a point symbol).
- ▶ If necessary you can also deselect elements that you already selected by clicking the element in the Graphics window again, but this time while pressing the CTRL key
- ▶ Alternative: Select the element in the List View window and press the **DEL** key

- ▶ Alternative: Click the icon to deselect all selected elements



- ▶ Save the selected machining positions to the clipboard of the control so that you can then insert them as a positioning block with cycle call in a Klartext program



- ▶ Alternative: Save the selected machining positions in a point file
- > The control displays a pop-up window in which you can select the target directory, a file name, and the file type.

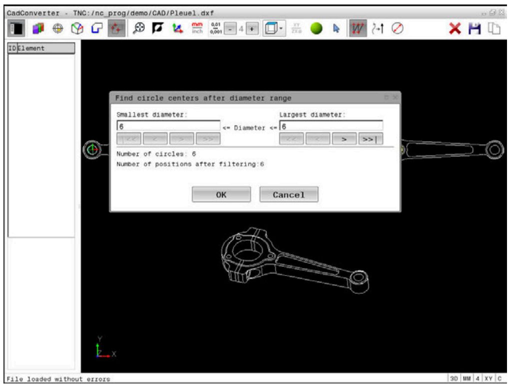
ENT

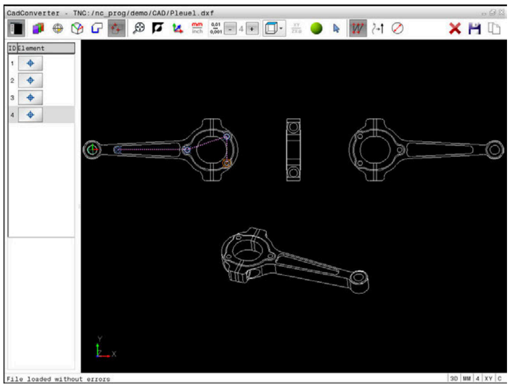
- ▶ Confirm the entry
- > The control saves the contour program to the selected directory.

- ▶ If you want to select more machining positions, press the Cancel Selected Elements icon and select as described above

Filter settings

After you have used the quick selection function to mark hole positions, a pop-up window appears in which the smallest diameter found is to the left and the largest diameter to the right. With the buttons just below the diameter display you can adjust the diameter so that you can load the hole diameters that you want.





The following buttons are available:

Icon	Filter setting of smallest diameter
¦<<	Display the smallest diameter found (default setting)
<	Display the next smaller diameter found
>	Display the next larger diameter found
>>	Display the largest diameter found. The control sets the filter for the smallest diameter to the value set for the largest diameter

Icon	Filter setting of largest diameter
<<	Display the smallest diameter found. The control sets the filter for the largest diameter to the value set for the smallest diameter
<	Display the next smaller diameter found
>	Display the next larger diameter found
>>¦	Display the largest diameter found (default setting)

You can display the tool paths via the **SHOW TOOL PATH** icon.

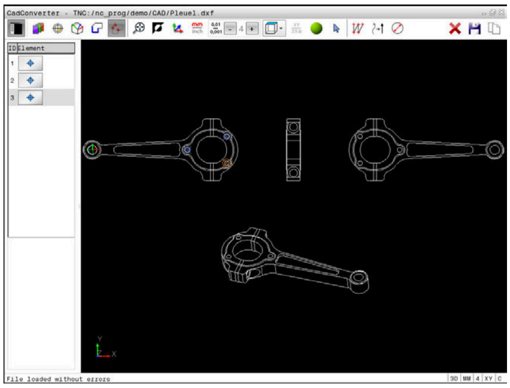
Further information: "Basic settings", Page 471

Element information

In the Element Information window, the control displays the coordinates of the machining position that you last selected in the List View window or Graphics window by clicking on the mouse.

You can also use the mouse to change the graphic display. The following functions are available:

- To rotate the model shown in three dimensions, hold down the right mouse button and move the mouse
- To shift the model shown, hold the center mouse button or mouse wheel down and move the mouse
- To zoom in on a certain area, mark a zoom area by holding the left mouse button down
- > After you release the left mouse button, the control zooms in on the defined area.
- To rapidly magnify or reduce any area, rotate the mouse wheel backwards or forwards
- To return to the standard display, press the shift key and simultaneously double-click with the right mouse button. The rotation angle is maintained if you only double-click with the right mouse button



13

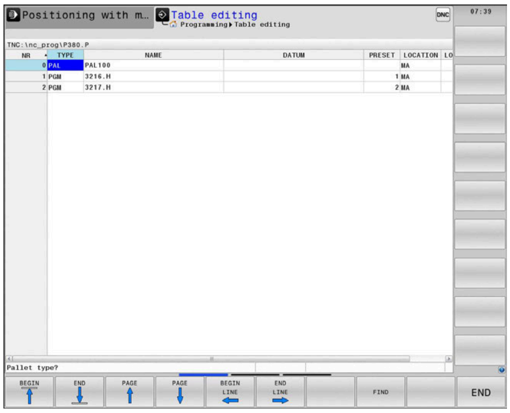
Pallets

13.1 Pallet management

Application

Refer to your machine manual!

Pallet table management is a machine-dependent function. The standard functional range is described below.



Pallet tables (**.p**) are mainly used in machining centers with pallet changers. The pallet tables call the different pallets (PAL), fixtures (FIX) optionally, and the associated NC programs (PGM). The pallet tables activate all defined presets and datum tables.

Without a pallet changer you can use pallet tables to process NC programs with different presets in sequence with just one press of **NC Start**.

The file name of a pallet table must always begin with a letter.

Columns of the pallet table

The machine tool builder defines a pallet table prototype that opens automatically when you create a pallet table.

The prototype can include the following columns:

Column	Meaning	Field type
NR	The control creates the entry automatically. The entry is required for the input field **Line number** of the **BLOCK SCAN** function.	Mandatory field
TYPE	The control differentiates between the following entries ■ **PAL** Pallet ■ **FIX** Fixture ■ **PGM** NC program Select the entries using the **ENT** key and the arrow keys or by soft key.	Mandatory field
NAME	File name The machine tool builder specifies the names for pallets and fixtures, if applicable, whereas you define program names. You must specify the complete path if the NC program is not saved in the directory of the pallet table.	Mandatory field
DATUM	Datum You must specify the complete path if the datum table is not saved in the directory of the pallet table. You activate datums from a datum table in the NC program using Cycle 7.	Optional field This entry is only required if a datum table is used.
PRESET	Workpiece preset Enter the preset number of the workpiece.	Optional field

Column	Meaning	Field type
LOCATION	Location of the pallet The entry **MA** indicates that there is a pallet or fixture in the working space of the machine and can be machined. Press the **ENT** key to enter **MA**. Press the **NO ENT** key to remove the entry and thus suppress machining.	Optional field If the column exists, the entry is mandatory.
LOCK	Line locked Using an * you can exclude the line of the pallet table from processing. Press the **ENT** key to identify the line with the entry *. Press the **NO ENT** key to cancel the lock. You can lock the execution for individual NC programs, fixtures or entire pallets. Unlocked lines (e.g. PGM) in a locked pallet are also not executed.	Optional field
PALPRES	Number of the pallet preset	Optional field This entry is only required if pallet presets are used.
W-STATUS	Execution status	Optional field This entry is only required for tool-oriented machining.
METHOD	Machining method	Optional field This entry is only required for tool-oriented machining.
CTID	ID for mid-program startup	Optional field This entry is only required for tool-oriented machining.
SP-X, SP-Y, SP-Z	Clearance height in the linear axes X, Y, and Z	Optional field
SP-A, SP-B, SP-C	Clearance height in the rotary axes A, B, and C	Optional field
SP-U, SP-V, SP-W	Clearance height in the parallel axes U, V, and W	Optional field
DOC	Comment	Optional field

You can remove the **LOCATION** column if you are only using pallet tables in which the control is to machine all lines.

Further information: "Inserting or deleting columns", Page 495

Editing a pallet table

When you create a new pallet table, it is empty at first. Using the soft keys, you can insert and edit lines.

Soft key	Editing function
BEGIN	Select the table start
END	Select the table end
PAGE	Select the previous page in the table
PAGE	Select the next page in the table
INSERT LINE	Insert as last line in the table
DELETE LINE	Delete the last line in the table
APPEND N LINES AT END	Add several lines at end of table
COPY FIELD	Copy the current value
PASTE FIELD	Insert the copied value
BEGIN LINE	Select beginning of line
END LINE	Select end of line
FIND	Find text or value
SORT/ HIDE COLUMNS	Sort or hide table columns
EDIT CURRENT FIELD	Edit the current field
SORT	Sort by column contents
MORE FUNCTIONS	Miscellaneous functions, e.g. saving
SELECT	Open file path selection

Selecting a pallet table

Proceed as follows to select a pallet table or create a new pallet table:

- Switch to the **Programming** mode or a program run mode

PGM MGT

- Press the **PGM MGT** key

If no pallet tables are shown:

SELECT TYPE

- Press the **SELECT TYPE** soft key
- Press the **SHOW ALL** soft key
- Select a pallet table with the arrow keys, or enter a name for a new pallet table (**.p**)

ENT

- Press the **ENT** key

> You can select either a list view or form view using the **Screen Layout** key.

Inserting or deleting columns

> This function is not enabled until the code number **555343** is entered.

Depending on the configuration, a newly created pallet table may not contain all columns. For tool-oriented machining, for example, you need columns that you have to insert first.

Proceed as follows to insert a column in an empty pallet table:

- Open the pallet table

MORE FUNCTIONS

- Press the **MORE FUNCTIONS** soft key

EDIT FORMAT

- Press the **EDIT FORMAT** soft key
- > The control opens a pop-up window displaying the available columns
- Using the arrow keys, select the desired column.

INSERT COLUMN

- Press the **INSERT COLUMN** soft key

ENT

- Press the **ENT** key

You can remove the column with the **DELETE COLUMN** soft key.

Fundamentals of tool-oriented machining

Application

Refer to your machine manual!

Tool-oriented machining is a machine-dependent function. The standard functional range is described below.

Tool-oriented machining allows you to machine several workpieces together even on a machine without pallet changer, which reduces tool-change times.

Limitation

NOTICE

Danger of collision!

Not all pallet tables and NC programs are suitable for tool-oriented machining. With tool-oriented machining, the control no longer executes the NC programs continuously, but divides them at the tool calls. The division of the NC programs allows functions that were not reset to be effective across programs (machine states). This leads to a danger of collision during machining!

- Consider the stated limitations
- Adapt pallet tables and NC programs to the tool-oriented machining
  - Reprogram the program information after each tool in every NC program (e.g. **M3** or **M4**).
  - Reset special functions and miscellaneous functions before each tool in every NC program (e. g. **Tilt the working plane** or **M138**)
- Carefully test the pallet table and associated NC programs in the **Program run, single block** operating mode

The following functions are not permitted:

- FUNCTION TCPM, M128
- M144
- M101
- M118
- Changing the pallet preset

The following functions require special attention, particularly for mid-program startup:

- Changing the machine statuses with a miscellaneous function (e.g. M13)
- Writing to the configuration (e.g. WRITE KINEMATICS)
- Traverse range switchover
- Cycle 32 Tolerance
- Cycle 800
- Tilting the working plane

Pallet table columns for tool-oriented machining

Unless the machine tool builder has made a different configuration, you need the following additional columns for tool-oriented machining:

Column	Meaning
W-STATUS	The machining status defines the machining progress. Enter BLANK for an unmachined (raw) workpiece. The control changes this entry automatically during machining. The control differentiates between the following entries ■ BLANK / no entry: Workpiece blank, requires machining ■ INCOMPLETE: Partly machined, requires further machining ■ ENDED: Machined completely, no further machining required ■ EMPTY: Empty space, no machining required ■ SKIP: Skip machining
METHOD	Indicates the machining method Tool-oriented machining is also possible with a combination of pallet fixtures, but not for multiple pallets. The control differentiates between the following entries ■ WPO: Workpiece oriented (standard) ■ TO: Tool oriented (first workpiece) ■ CTO: Tool oriented (further workpieces)
CTID	The control automatically generates the ID number for mid-program startup with block scan. If you delete or change the entry, mid-program startup is no longer possible.
SP-X, SP-Y, SP-Z, SP-A, SP-B, SP-C, SP-U, SP-V, SP-W	The entry for the clearance height in the existing axes is optional. You can enter safety positions for the axes. The control only approaches these positions if the machine tool builder processes them in the NC macros.

13.2 Batch Process Manager (option 154)

Application

Refer to your machine manual!

Your machine tool builder configures and enables the **Batch Process Manager** function.

The **Batch Process Manager** enables you to plan production orders on a machine tool.

You save the planned NC programs in a job list. You use the **Batch Process Manager** to open the job list.

The following information is displayed:

- Whether the NC program is free of errors
- Run time of the NC programs
- Availability of the tools
- Times at which manual interventions in the machine are required

The tool usage test function has to be enabled and switched on to ensure you get all information!

Further information: User's Manual for Setup, Testing and Running NC Programs

Fundamentals

The **Batch Process Manager** is available in the following operating modes:

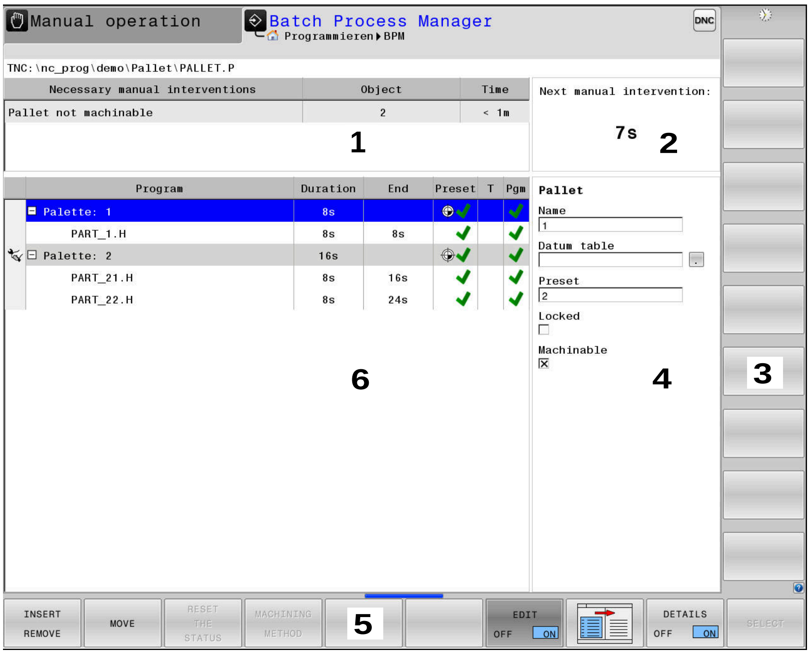
- **Programming**
- **Program run, single block**
- **Program run, full sequence**

In the **Programming** operating mode, you can create and edit the job list.

The job list is executed in the **Program run, single block** and **Program run, full sequence** operating modes. Changes are only possible to a limited extent.

Screen display

When you open the **Batch Process Manager** in the **Programming** operating mode, the following screen layout is displayed:



1	Displays all required manual interventions
2	Displays the next manual intervention
3	Displays the current soft keys provided by the machine tool builder if available
4	Shows the editable entries in the line highlighted in blue
5	Displays the current soft keys
6	Displays the job list

Columns of the job list

Column	Meaning
No column name	Status of the **Pallet**, **Clamping**, or **Program**
Program	Name or path of the **Pallet**, **Clamping**, or **Program**
Duration	Run time in seconds This column is only shown if you have a 19-inch screen.
End	End of the run time ■ Time in **Programming** operating mode ■ Actual time in **Program run, single block** and **Program run, full sequence** operating modes
Preset	Status of the workpiece preset
T	Status of the inserted tools
Pgm	Status of the NC program
Sts	Machining status

The status of the **Pallet**, **Clamping**, and **Program** is shown by means of icons in the first column.

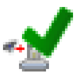
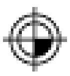
The icons have the following meanings:

Icon	Meaning
	The **Pallet**, **Clamping**, or **Program** is locked
	The **Pallet** or **Clamping** is not enabled for machining
	This line is currently being processed in **Program run, single block** or **Program run, full sequence** and cannot be edited
	The program was interrupted manually in this line

In the **Program** column, the machining method is indicated by icons.
The icons have the following meanings:

Icon	Meaning
No icon	Workpiece-oriented machining
	Tool-oriented machining ■ Start ■ End

The status is indicated by icons in the **Preset**, **T** and **Pgm** columns.
The icons have the following meanings:

Icon	Meaning
	Test completed
	Test completed Program simulation with active **Dynamic Collision Monitoring (DCM)** (Option 40)
	Test failed, e.g. because of expired tool life, danger of collision
	Test not yet completed
	Incorrect program structure, e.g.: pallet does not contain subordinate programs
	Workpiece preset is defined
	Check input You can either assign a workpiece preset to the pallet or to all subordinate NC programs.

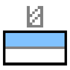
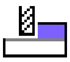
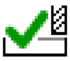
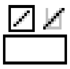
Operating notes:

- In **Programming** operating mode, the **T** column is always empty, because the control first checks the status in the **Program run, single block** and **Program run, full sequence** operating modes
- If the tool usage test function is not enabled or switched on on your machine, no icon is shown in the **Pgm** column

  Further information: User's Manual for Setup, Testing and Running NC Programs

In the **Sts** columns, the machining status is indicated by icons.

The icons have the following meanings:

Icon	Meaning
	Workpiece blank, requires machining
	Partly machined, requires further machining
	Machined completely, no further machining required
	Skip machining

Operating notes:

- The machining status is automatically adjusted during machining
- The **Sts** column is only shown in the **Batch Process Manager** if the pallet table contains the **W STATUS** column

  Further information: User's Manual for Setup, Testing and Running NC Programs

Opening the Batch Process Manager

Refer to your machine manual!

In machine parameter **standardEditor** (no. 102902), your machine tool builder specifies the standard editor used by the control.

Programming operating mode

If the control does not open the pallet table (.p) in the Batch Process Manager as a job list, proceed as follows:

- Select the desired job list
- Shift the soft-key row

MORE FUNCTIONS
- Press the **MORE FUNCTIONS** soft key

SELECT EDITOR
- Press the **SELECT EDITOR** soft key
- > The control opens the **Select editor** pop-up window.
- Select **BPM-EDITOR**

ENT
- Confirm your entry with the **ENT** key

OK
- Alternative: Press the **OK** soft key
- > The control opens the job list in the **Batch Process Manager**.

Program run, single block and Program run, full sequence operating modes

If the control does not open the pallet table (.p) in the Batch Process Manager as a job list, proceed as follows:

- Press the **Screen layout** key

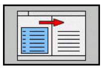
BPM
- Press the **BPM** key
- > The control opens the job list in the **Batch Process Manager**.

Soft keys

The following soft keys are available:

> Refer to your machine manual!
> The machine tool builder can configure his own soft keys.

Soft key	Function
DETAILS OFF ON	Collapse or expand tree structure
EDIT OFF ON	Edit opened job list
INSERT REMOVE	Shows the soft keys **INSERT BEFORE**, **INSERT AFTER** and **REMOVE**
MOVE	Move line
TAG	Select line

Soft key	Function
CANCEL THE MARKING	Cancel marking
INSERT BEFORE	Insert a new **Pallet**, **Clamping** or **Program** before the cursor position
INSERT AFTER	Insert a new **Pallet**, **Clamping**, or **Program** after the cursor position
REMOVE	Delete line or block
	Switch active windows
SELECT	Select possible entries from a pop-up window
RESET THE STATUS	Reset the machining status to workpiece blank
MACHINING METHOD	Select workpiece-oriented or tool-oriented machining
KOLLISIONS PRÜFUNG	Perform collision checking (Option 40) **Further information:** "Dynamic Collision Monitoring (option 40)", Page 356
KOLLISIONS PRÜFUNG ABBRECHEN	Abort collision checking (Option 40)
EINGRIFFE AUS EIN	Collapse or expand necessary manual interventions
TOOL MANAGEMENT	Open Extended Tool Management
INTERNAL STOP	Interrupt machining

Operating notes:

- The **TOOL MANAGEMENT**, **COLLISION CHECKING**, **ABORT COLLISION MONITORING** , and **INTERNAL STOP** soft keys are only available in the **Program run, single block** and **Program run, full sequence** operating modes.
- If the pallet table contains the **W STATUS** column, the **RESET THE STATUS** soft key is available.
- If the pallet table contains the **W STATUS**, **METHOD** and **CTID** columns, the **MACHINING METHOD** soft key is available.

Further information: User's Manual for Setup, Testing and Running NC Programs

Creating a job list

You can only create a new job list in the file manager.

> The file name of a job list must always begin with a letter.

- Press the **Programming** key
- Press the **PGM MGT** key
- > The control opens the file manager.
- Press the **NEW FILE** soft key
- Enter the file name with extension (**.p**)
- Confirm with the **ENT** key
- > The control opens an empty job list in the **Batch Process Manager**.
- Press the **INSERT REMOVE** soft key
- Press the **INSERT AFTER** soft key
- > The control displays the various types on the right-hand side.
- Select the desired type
  - **Pallet**
  - **Clamping**
  - **Program**
- > The control inserts an empty line in the job list.
- > The control shows the selected type on the right-hand side.
- Define the entries
  - **Name**: Enter the name directly or select one by means of the pop-up window, if there is one
  - **Datum table**: Enter the datum directly, where applicable, or select one by means of the pop-up window
  - **Preset**: Enter the workpiece preset directly, where applicable
  - **Locked**: The selected line is excluded from machining
  - **Machinable**: The selected line is enabled for machining
- Confirm your entries by pressing the **ENT** key.
- Repeat the steps if required
- Press the **EDIT** soft key

Editing a job list

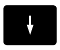
You can edit a job list in the **Programming**, **Program run, single block**, and **Program run, full sequence** operating modes.

> Operating notes:
>
> - If a job list is selected in the **Program run, single block** or **Program run, full sequence** operating mode, it is not possible to edit the job list in the **Programming** operating mode.
> - The possibilities of changing a job list during machining are limited, because the control defines a protected area.
> - NC programs in the protected area are shown in light gray.
> - If you edit the job list, the Collision checking completed status is reset to Check completed.

Proceed as follows to edit a line in the job list in the **Batch Process Manager**:

▶ Open the desired job list

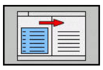
▶ Press the **EDIT** soft key



▶ Place the cursor on the desired line, e.g. **Pallet**

> The control displays the selected line in blue.

> The control displays the editable entries on the right-hand side.



▶ Press the **CHANGE WINDOW** soft key if required

> The control switches the active window.

▶ The following entries can be changed:

- **Name**
- **Datum table**
- **Preset**
- **Locked**
- **Machinable**

▶ Confirm the edited entries by pressing the **ENT** key.

> The control adopts the changes.

▶ Press the **EDIT** soft key

Proceed as follows to move a line in the job list in the **Batch Process Manager**:

- ▶ Open the desired job list

- ▶ Press the **EDIT** soft key

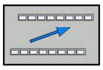
- ▶ Place the cursor on the desired line, e.g. **Program**
- > The control displays the selected line in blue.

- ▶ Press the **MOVE** soft key

- ▶ Press the **TAG** soft key
- > The control highlights the line in which the cursor is positioned.

- ▶ Place the cursor on the desired position.
- > When the cursor is placed at a suitable position, the control shows the **INSERT BEFORE** and **INSERT AFTER** soft keys.

- ▶ Press the **INSERT BEFORE** soft key
- > The control inserts the line at the new position.



- ▶ Press the **GO BACK** soft key

- ▶ Press the **EDIT** soft key

14

Turning

14.1 Turning operations on milling machines (option 50)

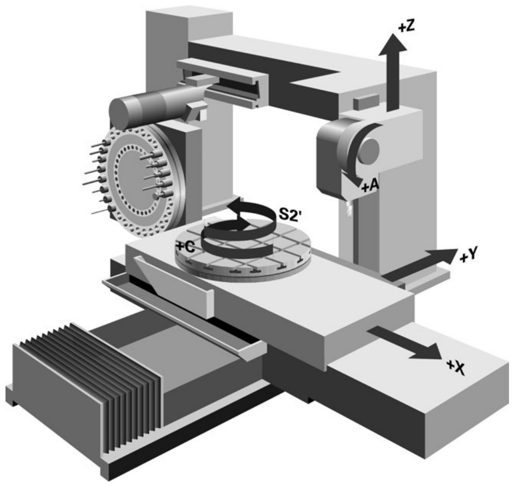
Introduction

Special types of milling machines allow performing both milling and drilling operations. A workpiece can thus be machined completely on one machine without rechucking, even if complex milling and turning applications are required.

Turning is a machining operation during which the workpiece rotates and thus performs the cutting movement. A fixed tool carries out infeed and feed movements.

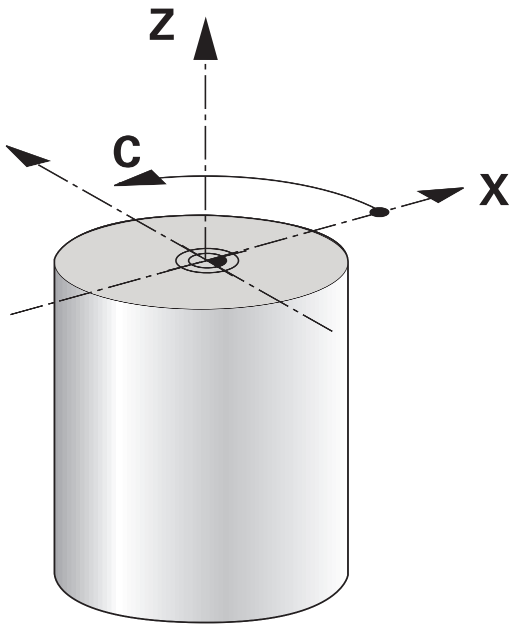
Turning applications, depending on machining direction and task, are subdivided into various production processes, e.g.

- Longitudinal turning
- Face turning
- Recess turning
- Thread cutting



The control offers you several cycles for each of the various production processes.

Further information: Cycle Programming User's Manual



On the control you can simply switch between milling and turning mode within the NC program. In turning mode, the rotary table serves as lathe spindle, whereas the milling spindle with the tool is fixed. This enables rotationally symmetric contours to be created. The preset must be in the center of the lathe spindle for this.

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

Further information: User's Manual for Setup, Testing and Running NC Programs

Different cycles are available for machining. These can also be used with additional swivel axes.

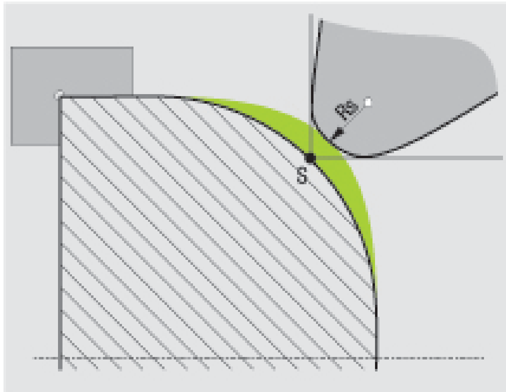
Further information: "Inclined turning", Page 525

Coordinate plane of turning operations

The assignment of the axes with turning is defined so that the X coordinates describe the diameter of the workpiece and the Z coordinates the longitudinal positions.

Programming is thus always done in the ZX coordinate plane. The machine axes to be used for the required movements depend on the respective machine kinematics and are determined by the machine manufacturer. This makes NC programs with turning functions largely exchangeable and independent of the machine model.

Tool radius compensation TRC

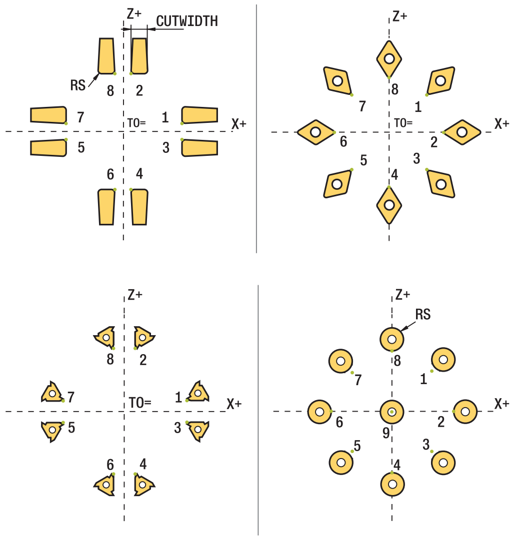


The tip of a lathe tool has a certain radius (**RS**). When machining tapers, chamfers and radii, this results in distortions on the contour because the programmed traverse paths refer to the theoretical tool tip S. TRC prevents the resulting deviations.

In the turning cycles the control automatically carries out tool radius compensation. In specific traversing blocks and within programmed contours, activate TRC with **RL** or **RR**.

The control checks the cutting geometry with the point angle **P-ANGLE** and the setting angle **T-ANGLE**. Contour elements in the cycle are processed by the control only as far as this is possible with the specific tool.

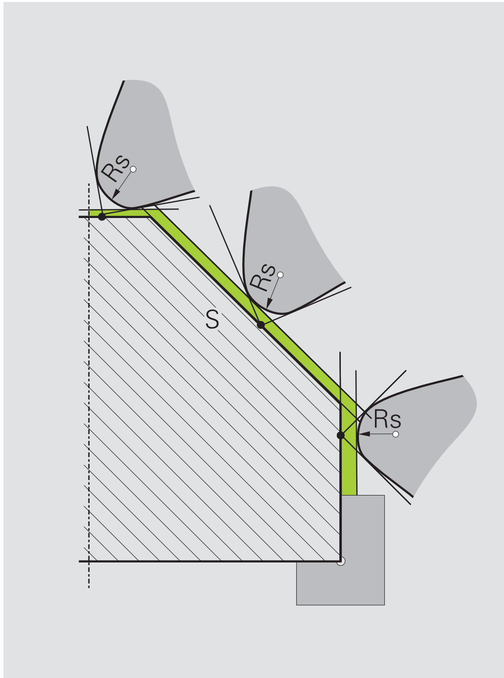
The control displays a warning when residual material is left behind due to the angle of the secondary cutting edges. You can suppress the warning with the machine parameter **suppressResMatlWar** (no. 201010).



> **Programming notes:**
> - The direction of the radius compensation is not clear when the tool-tip position (**TO=2, 4, 6, 8**) is neutral. In this case, TRC is only possible within fixed machining cycles.
>
>   The control can also run tool tip radius compensation during inclined processing.
>
>   Active miscellaneous functions limit the possibilities here:
>   - With **M128** tool-tip radius compensation is possible only in combination with machining cycles
>   - **M144** or **FUNCTION TCPM** with **REFPNT TIP-CENTER** also allows tool-tip radius compensation with all positioning blocks, e.g. with **RL**/**RR**

Theoretical tool tip

The theoretical tool tip is effective in the tool coordinate system. When the tool is inclined, the position of the tool tip rotates with the tool.



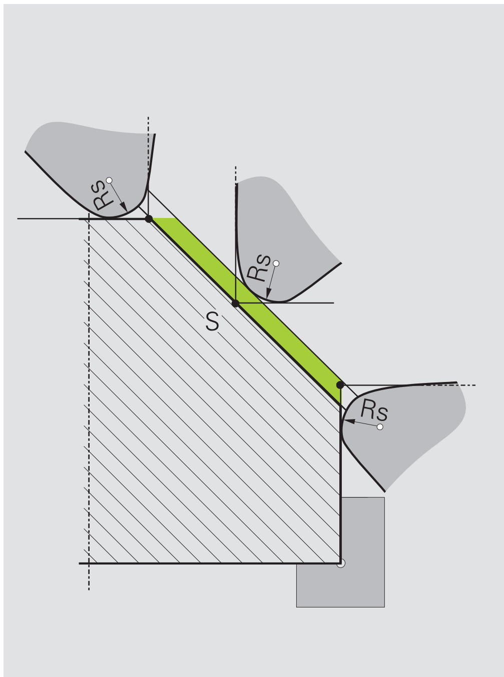
Virtual tool tip

Use **FUNCTION TCPM** with the selection **REFPNT TIP-CENTER** to activate the virtual tool tip. Correct tool data are the prerequisite for calculating the virtual tool tip.

The virtual tool tip is effective in the workpiece coordinate system. When the tool is inclined, the virtual tool tip remains unchanged as long as the tool orientation **TO** is the same. The control automatically switches the status display **TO** and thus also the virtual too tip if the tool leaves the angle range valid for **TO 1**, for example.

The virtual tool tip enables you to perform inclined paraxial longitudinal and transverse machining operations with high contour accuracy even without radius compensation.

Further information: "Simultaneous turning", Page 527



14.2 Basic functions (option 50)

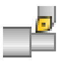
Switching between milling and turning mode

> Refer to your machine manual!
> The machine tool builder configures and enables turning and switchover of the machining modes.

To switch between milling and turning operations, you must switch to the specific mode.

You can switch these operating modes with the NC functions **FUNCTION MODE TURN** and **FUNCTION MODE MILL**.

If turning mode is active, the control displays a corresponding icon in the status display.

Icon	Mode
	Turning mode active: **FUNCTION MODE TURN**
No icon	Milling mode active: **FUNCTION MODE MILL**

When the operating modes are switched, the control executes a macro that defines the machine-specific settings for the specific operating mode. With the NC functions **FUNCTION MODE TURN** and **FUNCTION MODE MILL** you can activate a machine kinematic model that the machine tool builder has defined and saved in the macro.

NOTICE

Caution: Significant property damage!

Very high physical forces are generated during turning, for example by high rotational speeds and heavy or unbalanced workpieces. Incorrect machining parameters, neglected unbalances or improper fixtures lead to an increased risk of accidents during machining!

- Clamp the workpiece in the spindle center
- Clamp workpiece securely
- Program low spindle speeds (increase as required)
- Limit the spindle speed (increase as required)
- Eliminate unbalance (calibrate)

Programming notes:

- If the **Tilt working plane** or **TCPM** functions are active, you cannot switch the operating mode.
- In turning mode, no coordinate conversion cycles are permitted except for the datum shift.
- The orientation of the tool spindle (spindle angle) depends on the machining direction. The tool tip is aligned to the center of the turning spindle for outside machining. For inside machining, the tool points away from the center of the turning spindle.
- The direction of spindle rotation must be adapted when the machining direction (outside/inside machining) is changed.
- During turning, the cutting edge and the center of the turning spindle must be at the same level. During turning, the tool therefore has to be pre-positioned to the Y coordinate of the turning-spindle center.
- By means of M138, you can select the rotary axes for M128 and TCPM.

Operating notes:

- The preset must be in the center of the turning spindle in turning mode.
- In turning mode, diameter values are displayed on the X axis position display. The control then shows an additional diameter symbol.
- In turning mode, the spindle potentiometer is active for the turning spindle (rotary table).
- In turning mode you can use all manual touch probe cycles, except the **Probe corner** and **Probe plane** cycles. In turning mode, the measured values of the X axis equal diameter values.
- You can also use the smartSelect function to define the turning functions.
  Further information: "Overview of special functions", Page 352

Specifying the machining mode

SPEC FCT

- Show the soft-key row with special functions

FUNCTION MODE

- Press the **FUNCTION MODE** soft key

TURN

- Function for machining mode: Press the **TURN** (Turning) or **MILL** (Milling) soft key

If the machine tool builder has enabled kinematics selection, proceed as follows:

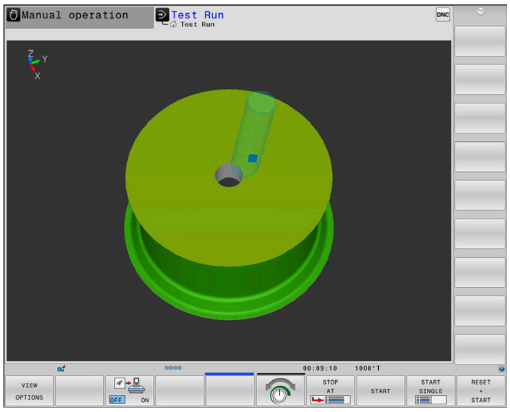
SELECT

- Press the **SELECT KINEMATICS** soft key
- Select the desired kinematics

Example

11 FUNCTION MODE TURN "AC_TABLE"	Activate turning mode
12 FUNCTION MODE TURN	Activate turning mode
13 FUNCTION MODE MILL "B_HEAD"	Activate milling mode

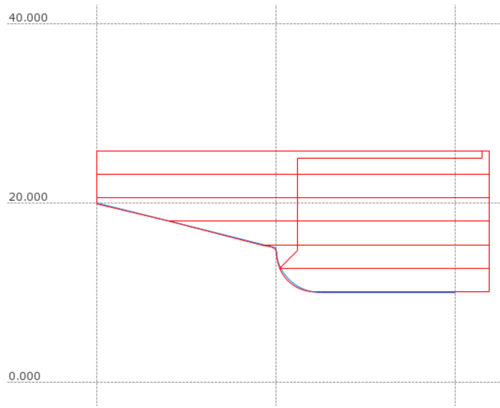
Graphic display of turning operations



You can simulate turning operations in **Test Run** mode. The requirement for this is a workpiece blank definition suitable for the turning process and option number 20.

> The machining times determined using the graphic simulation do not correspond to the actual machining times. Reasons for this during combined milling-turning operations include the switching of operating modes.

Graphic display in the Programming mode of operation



You can graphically simulate turning operations with the line graphic in **Programming** operating mode. To display the traverse movements in turning mode in **Programming** operating mode, change the layout using the soft keys.

Further information: "Generating a graphic for an existing NC program", Page 207

The standard assignment of the axes with turning is defined so that the X coordinates describe the diameter of the workpiece and the Z coordinates the longitudinal positions.

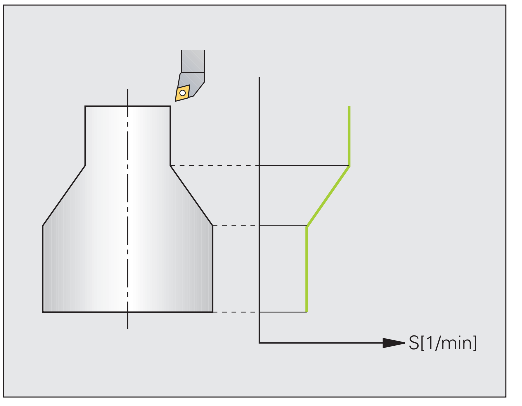
Even if the turning operation takes place in a two-dimensional plane (Z and X coordinates), you have to program the Y values for a rectangular blank in the definition of the workpiece blank.

Example. Rectangular blank

0 BEGIN PGM BLK MM	
1 BLK FORM 0.1Y X+0 Y-1 Z-50	Workpiece blank definition
2 BLK FORM 0.2 X+87 Y+1 Z+2	
3 TOOL CALL 12	Tool call
4 M140 MB MAX	Retract the tool
5 FUNCTION MODE TURN	Activate Turning mode

Programming the spindle speed

⚙	Refer to your machine manual! If you machine at constant cutting speed, the selected gear range limits the possible spindle speed range. The possible gear ranges (if applicable) depend on your machine.



With turning you can machine both at constant spindle speed and constant cutting speed.

If you machine at constant cutting speed **VCONST:ON**, the control modifies the speed according to the distance of the tool tip to the center of the turning spindle. For positioning movements toward the center of rotation, the control increases the table speed; for movements away from the center of rotation, it reduces the table speed.

For processing with constant spindle speed **VCONST:Off**, speed is independent of the tool position.

Use **FUNCTION TURNDATA SPIN** to define the speed. The control provides the following input parameters:

- VCONST: Constant cutting speed on/off (optional)
- VC: Cutting speed (optional)
- S: Nominal speed if no constant cutting speed is active (optional)
- S MAX: Maximum speed with constant cutting speed (optional). Reset with S MAX 0
- GEARRANGE: Gear range for the turning spindle (optional)

Defining the speed:

SPEC FCT

- Show the soft-key row with special functions

TURNING PROGRAM FUNCTIONS

- Press the **TURNING PROGRAM FUNCTIONS** soft key

FUNCTION TURNDATA

- Press the **FUNCTION TURNDATA** soft key

TURNDATA SPIN

- Press the **TURNDATA SPIN** soft key.

VCONST: ON

- Select the function for speed entry: Press the **VCONST:** soft key

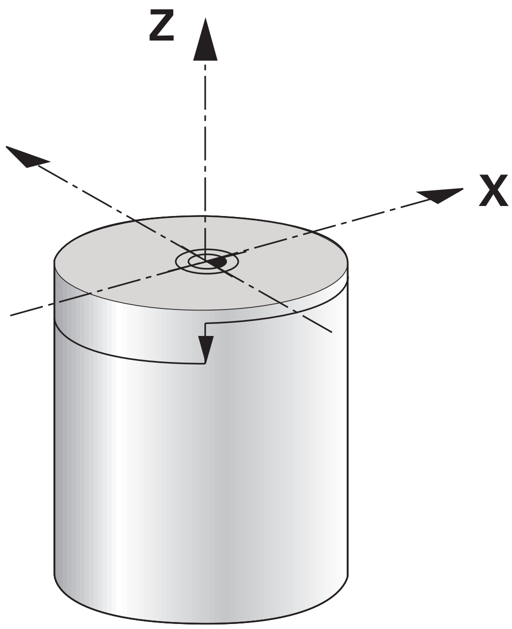
> Cycle 800 limits maximum speed with eccentric turning. The control restores a programmed limitation of the spindle speed after eccentric turning.
>
> To reset the speed limitation, program **FUNCTION TURNDATA SPIN SMAX0**.
>
> If the maximum speed is achieved the control displays **SMAX** instead of **S** in the status display.

Example

3 FUNCTION TURNDATA SPIN VCONST:ON VC:100 GEARRANGE:2	Definition of a constant cutting speed in gear range 2
3 FUNCTION TURNDATA SPIN VCONST:OFF S550	Definition of a constant spindle speed
...	

Feed rate

With turning, feed rates are often specified in millimeters per revolution. The control thus moves the tool at a defined value for every spindle rotation. The resulting contouring feed rate is thus dependent on the speed of the turning spindle. The control increases the feed rate at high spindle speeds and reduces it at low spindle speeds. This enables you to machine with uniform cutting depth and constant cutting force, thus achieving constant chip thickness



> During many turning operations, it is not possible to maintain constant surface speeds (**VCONST: ON**) because the maximum spindle speed is reached first. Use the machine parameter **facMinFeedTurnSMAX** (no. 201009) to define the behavior of the control after the maximum speed has been reached.

By default, the control interprets the programmed feed rate in millimeters per minute (mm/min). If you want to define the feed rate in millimeters per revolution (mm/1), you have to program **M136**. The control then interprets all subsequent feed rate specifications in mm/1 until **M136** is canceled.

M136 is effective modally at the beginning of the block and can be canceled with **M137**.

Example

10 L X+102 Z+2 R0 FMAX	Movement at rapid traverse
...	
15 L Z-10 F200	Movement at a feed rate of 200 mm/min
...	
19 M136	Feed rate in millimeters per revolution
20 L X+154 F0.2	Movement at a feed rate of 0.2 mm/1
...	

14.3 Turning program functions (option 50)

Tool compensation in the NC program

With **FUNCTION TURNDATA CORR** you can define additional compensation values for the active tool. In the **TURNDATA CORR FUNCTION** you can enter delta values for tool lengths in the X direction **DXL** and in the Z direction **DZL**. The compensation values have an additive effect on the compensation values from the turning tool table.

With **FUNCTION TURNDATA CORR-TCS** you can define a cutter radius oversize **DRS**. This enables you to program an equidistant contour oversize. **DCW** allows you to compensate the recessing width of a recessing tool.

FUNCTION TURNDATA CORR is always effective for the active tool. A renewed **TOOL CALL** deactivates compensation again. When you exit the NC program (e.g. with PGM MGT), the control automatically resets the compensation values.

When using the **FUNCTION TURNDATA CORR** function, you can define via soft keys where the tool compensation will be effective:

- **FUNCTION TURNDATA CORR-TCS**: The tool compensation is effective in the tool coordinate system
- **FUNCTION TURNDATA CORR-WPL**: The tool compensation is effective in the workpiece coordinate system

> Tool compensation **FUNCTION TURNDATA CORR-TCS** is always effective in the tool coordinate system, even during inclined machining.

> During interpolation turning the functions **FUNCTION TURNDATA CORR** and **FUNCTION TURNDATA CORR-TCS** do not have any effect.
>
> If you want to compensate a turning tool during interpolation turning (Cycle 292), compensation needs to be performed in the cycle or in the tool table.
>
> **Further information:** Cycle Programming User's Manual

Defining the tool compensation

To define the tool compensation in the NC program, proceed as follows:

SPEC FCT
- Press the **SPEC FCT** key

TURNING PROGRAM FUNCTIONS
- Press the **TURNING PROGRAM FUNCTIONS** soft key

FUNCTION TURNDATA
- Press the **FUNCTION TURNDATA** soft key

TURNDATA CORR
- Press the **TURNDATA CORR** soft key.

> As an alternative to tool compensation with **TURNDATA CORR**, you can use compensation tables.
>
> **Further information:** "Compensation table", Page 377

Example

```
21 FUNCTION TURNDATA CORR-TCS:Z/X DZL:0.1 DXL:0.05
...
```

Recessing and undercutting

Some cycles machine contours that you have written in a subprogram. You program these contours with path functions or FK functions. Further special contour elements are available to you for writing turning contours. In this way you can program recessing and undercutting as complete contour elements with a single NC block.

> Recessing and undercutting always reference a previously defined linear contour element.
>
> You can only use the recess and undercut elements GRV and UDC in contour subprograms that have been called by a turning cycle.
>
> **Further information:** Cycle Programming User's Manual

Various input options are available to you for defining undercuts and recesses. Some of these inputs have to be made (mandatory input), some can be skipped (optional input). The mandatory inputs are symbolized as such in the help graphics. In some elements you can select between two different definitions. The controls has soft keys with the corresponding selection possibilities.

Programming recessing and undercutting:

SPEC FCT
- Show the soft-key row with special functions

TURNING PROGRAM FUNCTIONS
- Press the **TURNING PROGRAM FUNCTIONS** soft key

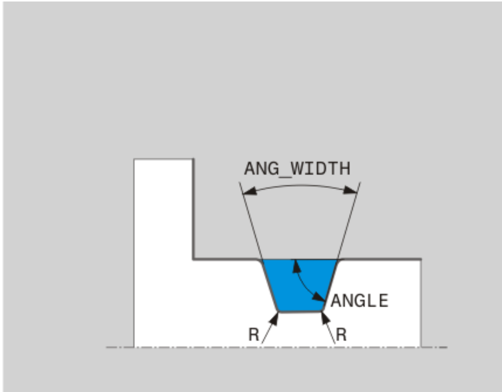
RECESS/ UNDERCUT
- Press the **RECESS/ UNDERCUT** soft key

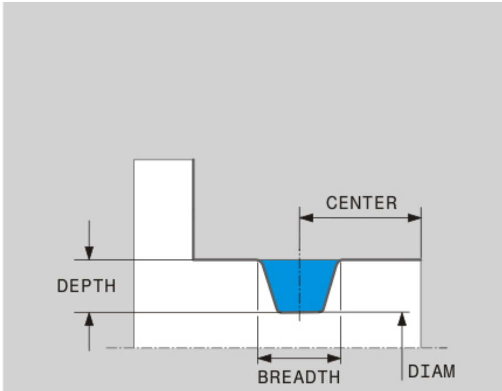
GRV
- Press the **GRV** (recess) or **UDC** (undercut) soft key

Programming recessing

Recessing is the machining of recesses in round components, usually for accommodation of locking rings and seals or as lubricating grooves. You can program recessing around the circumference or on the face end of the turned part. For this you have two separate contour elements:

- **GRV RADIAL**: Recess in circumference of component
- **GRV AXIAL**: Recess on face end of component





Input parameters in recessing GRV

Input parameters	Application	Input
CENTER	Center of recess	Required
R	Corner radius of both inner corners	Optional
DEPTH / **DIAM**	Recess depth (pay attention to the sign!) / diameter of recess base	Required
BREADTH	Recess width	Required
ANGLE / **ANG_WIDTH**	Edge angle / aperture angle of both edges	Optional
RND / **CHF**	Curve / chamfer corner of contour near to starting point	Optional
FAR_RND / **FAR_CHF**	Curve / chamfer corner of contour away from starting point	Optional

> The algebraic sign for the recess depth specifies the machining position (inside/outside machining) of the recess.
>
> Algebraic sign of recess depth for outside machining:
>
> - If the contour element is in the negative direction of the Z coordinate, use a negative sign
> - If the contour element is in the positive direction of the Z coordinate, use a positive sign
>
> Algebraic sign of recess depth for inside machining:
>
> - If the contour element is in the negative direction of the Z coordinate, use a positive sign
> - If the contour element is in the positive direction of the Z coordinate, use a negative sign

Example: Radial recess with depth=5, width=10, pos.= Z-15

```
21 L  X+40  Z+0
22 L  Z-30
23 GRV RADIAL CENTER-15 DEPTH-5 BREADTH10 CHF1 FAR_CHF1
24 L  X+60
```

Programming undercutting

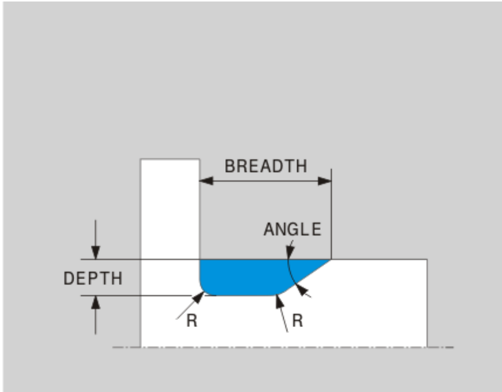
Undercutting is usually required for the flush connection of counterparts. In addition undercutting can help to reduce the notch effect at corners. Threads and fits are often machined with an undercut. You have various contour elements for defining the different undercuts:

- **UDC TYPE_E**: Undercut for cylindrical surface to be further processed in compliance with DIN 509
- **UDC TYPE_F**: Undercut for plan and cylindrical surface for further processing in compliance with DIN 509
- **UDC TYPE_H**: Undercut for more rounded transition in compliance with DIN 509
- **UDC TYPE_K**: Undercut in face and cylindrical surface
- **UDC TYPE_U**: Undercut in cylindrical surface
- **UDC THREAD**: Thread undercut in compliance with DIN 76

> The control always interprets undercuts as form elements in the longitudinal direction. No undercuts are possible in the plane direction.

Undercut DIN 509 UDC TYPE _E

Input parameters in undercut DIN 509 UDC TYPE_E



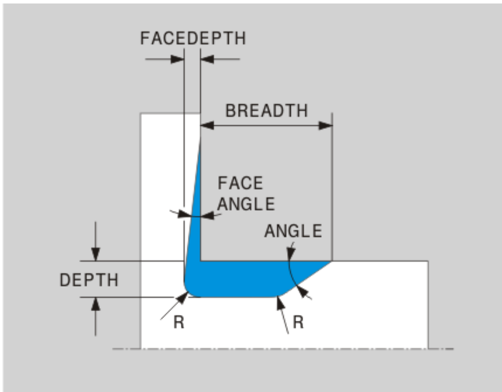
Input parameters	Application	Input
R	Corner radius of both inner corners	Optional
DEPTH	Undercut depth	Optional
BREADTH	Width of undercut	Optional
ANGLE	Undercut angle	Optional

Example: Undercut with depth = 2, width = 15

```
21 l  X+40 Z+0
22 l  Z-30
23 UDC TYPE_E R1 DEPTH2 BREADTH15
24 L  X+60
```

Undercut DIN 509 UDC TYPE_F

Input parameters in undercut DIN 509 UDC TYPE_F



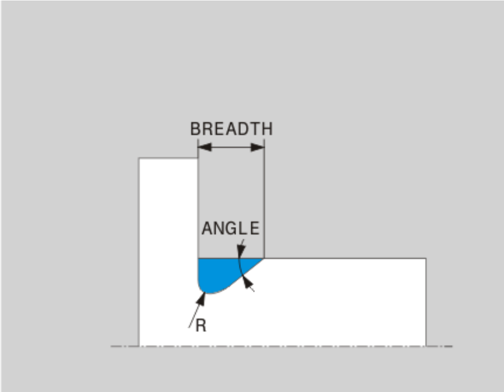
Input parameters	Application	Input
R	Corner radius of both inner corners	Optional
DEPTH	Undercut depth	Optional
BREADTH	Width of undercut	Optional
ANGLE	Undercut angle	Optional
FACEDEPTH	Depth of face	Optional
FACEANGLE	Contour angle of face	Optional

Example: Undercut form F with depth = 2, width = 15, depth of face = 1

```
21 L  X+40  Z+0
22 L  Z-30
23 UDC TYPE_F R1 DEPTH2 BREADTH15 FACEDEPTH1
24 L  X+60
```

Undercut DIN 509 UDC TYPE_H

Input parameters in undercut DIN 509 UDC TYPE_H



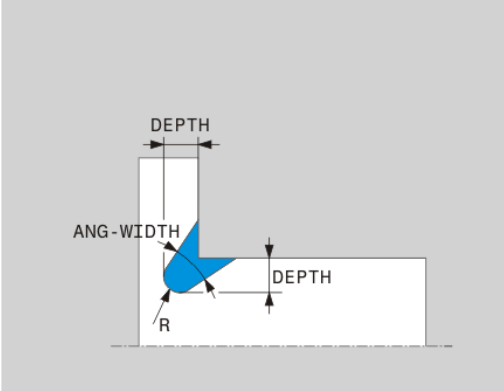
Input parameters	Application	Input
R	Corner radius of both inner corners	Required
BREADTH	Width of undercut	Required
ANGLE	Undercut angle	Required

Example: Undercut form H with depth = 2, width = 15, angle = 10°

```
21 L  X+40  Z+0
22 L  Z-30
23 UDC TYPE_H R1 BREADTH10 ANGLE10
24 L  X+60
```

Undercut UDC TYPE_K

Input parameters in undercut UDC TYPE_K



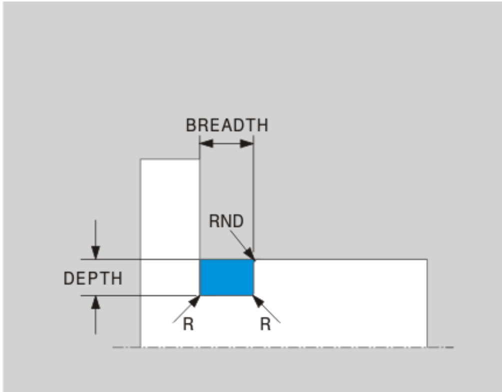
Input parameters	Application	Input
R	Corner radius of both inner corners	Required
DEPTH	Undercut depth (paraxial)	Required
ROT	Angle to longitudinal axis (default: 45°)	Optional
ANG_WIDTH	Opening angle of undercut	Required

Example: Undercut form K with depth = 2, width = 15, opening angle = 30°

```
21 L  X+40  Z+0
22 L  Z-30
23 UDC TYPE_K R1 DEPTH3 ANG_WIDTH30
24 L  X+60
```

Undercut UDC TYPE_U

Input parameters in undercut UDC TYPE_U



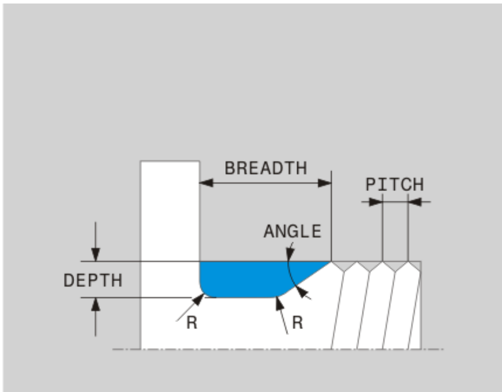
Input parameters	Application	Input
R	Corner radius of both inner corners	Required
DEPTH	Undercut depth	Required
BREADTH	Width of undercut	Required
RND / **CHF**	Curve / chamfer of outer corner	Required

Example: Undercut form U with depth = 3, width = 8

```
21 L  X+40  Z+0
22 L  Z-30
23 UDC TYPE_U R1 DEPTH3 BREADTH8 RND1
24 L  X+60
```

Undercut UDC THREAD

Input parameters in undercut DIN 76 UDC THREAD



Input parameters	Application	Input
PITCH	Thread pitch	Optional
R	Corner radius of both inner corners	Optional
DEPTH	Undercut depth	Optional
BREADTH	Width of undercut	Optional
ANGLE	Undercut angle	Optional

Example: Thread undercut according to DIN 76 with thread pitch = 2

```
21 L  X+40  Z+0
22 L  Z-30
23 UDC THREAD PITCH2
24 L  X+60
```

Blank form update TURNDATA BLANK

The **TURNDATA BLANK** function enables you to use the blank form update feature. The control detects the described contour and only then machines the residual material.

With **TURNDATA BLANK** you call a contour description used by the control as an updated workpiece blank.

Define the function TURNDATA BLANK as follows:

SPEC FCT
- Show the soft-key row with special functions

TURNING PROGRAM FUNCTIONS
- Press the **TURNING PROGRAM FUNCTIONS** soft key

FUNCTION TURNDATA
- Press the **FUNCTION TURNDATA** soft key

TURNDATA BLANK
- Press the **TURNDATA BLANK** soft key
- Press the soft key for the desired contour call

You can call the contour description in the following ways:

Soft key	Call
BLANK <FILE>	Contour description in an external NC program Call via file name
BLANK <FILE>=QS	Contour description in an external NC program Call via string parameter
BLANK LBL NR	Contour description in a subprogram Call via label number
BLANK LBL NAME	Contour description in a subprogram Call via label name
BLANK LBL QS	Contour description in a subprogram Call via string parameter

Deactivate blank form update

Deactivate blank form update as follows:

SPEC FCT
- Show the soft-key row with special functions

TURNING PROGRAM FUNCTIONS
- Press the **TURNING PROGRAM FUNCTIONS** soft key

FUNCTION TURNDATA
- Press the **FUNCTION TURNDATA** soft key

TURNDATA BLANK
- Press the **TURNDATA BLANK** soft key

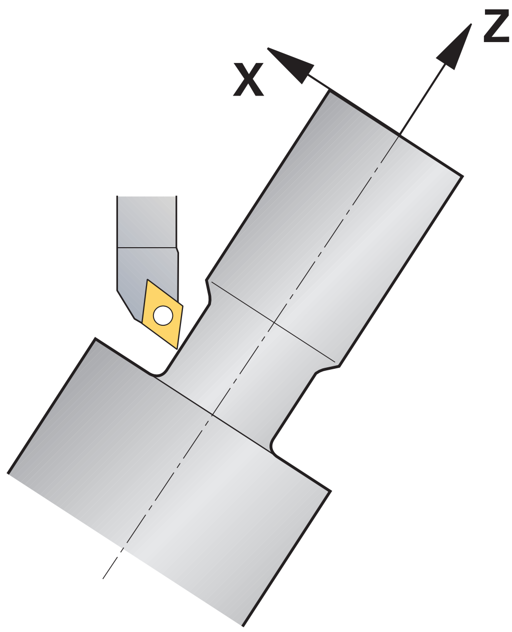
BLANK OFF
- Press the **BLANK OFF** soft key

Inclined turning

It may sometimes be necessary for you to bring the swivel axes into a specific position to machine a specific process. This can be necessary for example when you can only machine contour elements according to a specific position due to tool geometry.

The control offers the following methods of inclined turning:

- **M144**
- **M128**
- **FUNCTION TCPM** with **REFPNT TIP-CENTER**
- Cycle 800 **ADJUST XZ SYSTEM**
  Further information: Cycle Programming User's Manual



If you execute turning cycles with **M144**, **FUNCTION TCPM**, or **M128**, then the angles of the tool relative to the contour will change. The control automatically takes these modifications into account and therefore also monitors the inclined machining operation.

Programming notes:

- Recessing cycles and threading cycles can be run with inclined machining only if the tool is at a right angle (+90°, or –90°).
- Tool compensation **FUNCTION TURNDATA CORR-TCS** is always effective in the tool coordinate system, even during inclined machining.

M144

Inclining a swivel axis creates an offset from tool to tool. The function **M144** considers the position of the inclined axes and compensates this offset. In addition the function **M144** aligns the Z direction of the workpiece coordinate system to the direction of the centerline of the workpiece. If an inclined axis is a tilting table, meaning that the workpiece itself is inclined, the control performs traverse movements in the rotated workpiece coordinate system. If the inclined axis is a swivel head (meaning that the tool is inclined), the workpiece coordinate system is not rotated.

After inclining the swivel axis you may have to again pre-position the tool in the Y coordinate and orient the position of the tool tip with Cycle 800.

Example

...	
12 M144	Activate inclined machining
13 L A-25 R0 FMAX	Position swivel axis
14 CYCL DEF 800 ADJUST XZ SYSTEM	Workpiece coordinate system and align tool
Q497=+90 ;PRECESSION ANGLE	
Q498=+0 ;REVERSE TOOL	
Q530=+2 ;INCLINED MACHINING	
Q531=-25 ;ANGLE OF INCIDENCE	
Q532=750 ;FEED RATE	
Q533=+1 ;PREFERRED DIRECTION	
Q535=3 ;ECCENTRIC TURNING	
Q536=0 ;ECCENTRIC W/O STOP	
15 L X+165 Y+0 R0 FMAX	Pre-positioning the tool
16 L Z+2 R0 FMAX	Tool at starting position
...	Machining with inclined axis

M128

Alternately, you can use the **M128** function The effect is the same, but the following limitation applies here: if you activate inclined machining with M128 then tool-tip radius compensation without a cycle, i.e. in traversing blocks with **RL**/**RR**, is not possible. If you activate inclined machining via **M144** or **FUNCTION TCPM** with **REFPNT TIP-CENTER** then this limitation does not apply.

FUNCTION TCPM with REFPNT TIP-CENTER

Use **FUNCTION TCPM** with the selection **REFPNT TIP-CENTER** to activate the virtual tool tip. If you activate inclined machining with **FUNCTION TCPM** with **REFPNT TIP-CENTER**, then tool-tip radius compensation is also possible without a cycle; that is, in traversing blocks with **RL**/**RR**.

In **Manual operation** mode, you can also perform inclined turning if you activate **FUNCTION TCPM** with the selection **REFPNT TIP-CENTER** in, for example, the **Positioning w/ Manual Data Input** operating mode.

Simultaneous turning

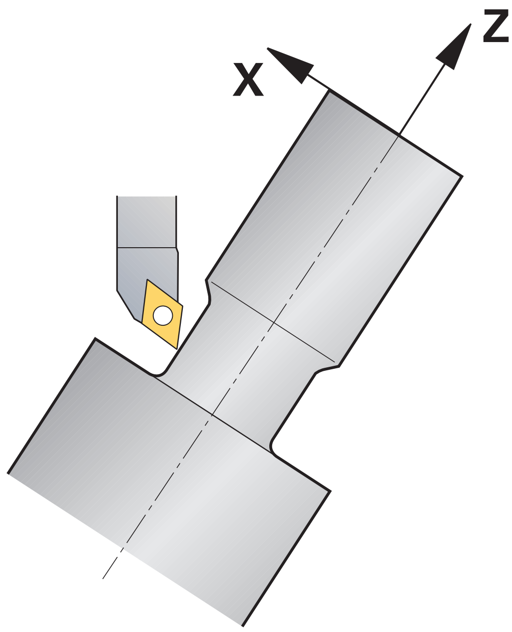
You can combine the turning operation with function **M128** or **FUNCTION TCPM** and **REFPNT TIP-CENTER**. This enables you to manufacture contours in one cut, for which you have to change the inclination angle (simultaneous machining).

The simultaneous turning contour is a turning contour for which a rotary axis whose inclination does not violate the contour can be programmed on **CP** polar circles and **L** linear blocks. Collisions with lateral cutting edges or holders are not prevented. This makes it possible to finish contours with one tool in a continuous movement, even though different sections of the contour are accessible only in different tool inclinations.

In the NC program you define how the rotary axis has to be inclined to reach the different contour parts without collisions.

Use the cutter radius oversize **DRS** to leave an equidistant oversize on the contour.

Use **FUNCTION TCPM** and **REFPNT TIP-CENTER** to measure the theoretical tool tip of the turning tools being used for this.



Procedure

To write a simultaneous program, proceed as follows:

- Activate turning mode
- Insert a turning tool.
- Adjust the coordinate system with Cycle 800
- Activate **FUNCTION TCPM** with **REFPNT TIP-CENTER**
- Activate radius compensation with RL / RRG41/G42
- Program simultaneous turning contour
- Cancel radius compensation with a departure block or R0
- Reset **FUNCTION TCPM**

Example

0 BEGIN PGM TURNSIMULTAN MM	
...	
12 FUNCTION MODE TURN	Activate turning mode
13 TOOL CALL "TURN_FINISH"	Insert a turning tool
14 FUNCTION TURNDATA SPIN VCONST:OFF S500	
15 M140 MB MAX	
16 CYCL DEF 800 ADJUST XZ SYSTEM	Adapt the coordinate system
Q497=+90 ;PRECESSION ANGLE	
Q498=+0 ;REVERSE TOOL	
Q530=+0 ;INCLINED MACHINING	
Q531=+0 ;ANGLE OF INCIDENCE	
Q532= MAX ;FEED RATE	
Q533=+0 ;PREFERRED DIRECTION	
Q535=+3 ;ECCENTRIC TURNING	
Q536=+0 ;ECCENTRIC W/O STOP	
17 FUNCTION TCPM F TCP AXIS POS PATHCTRL AXIS REFPNT TIP-CENTER	Activate FUNCTION TCPM
18 FUNCTION TURNDATA CORR-TCS:Z/X DRS:-0.1	
19 L X+100 Y+0 Z+10 R0 FMAX M304	
20 L X+45 RR FMAX	Activate radius compensation with RR
...	
26 L Z-12.5 A-75	Program simultaneous turning contour
27 L Z-15	
28 CC X+69 Z-20	
29 CP PA-90 A-45 DR-	
30 CP PA-180 A+0 DR-	
...	
47 L X+100 Z-45 R0 FMAX	Cancel radius compensation with R0
48 FUNCTION RESET TCPM	Reset FUNCTION TCPM
49 FUNCTION MODE MILL	
...	
71 END PGM TURNSIMULTAN MM	

M128

Alternately, you can use the **M128** function for simultaneous turning

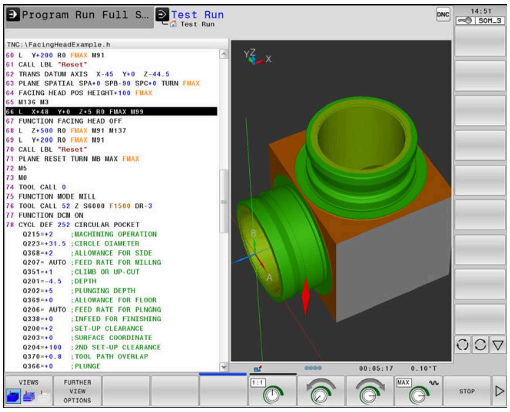
The following constraints apply for M128:

- Only for NC programs programmed on the path of the tool center.
- Only for button turning tools with TO 9
- The tool must be measured at the center of the tool-tip radius

Using a facing slide

Application

> Refer to your machine manual!
>
> This function must be enabled and adapted by the machine tool builder.



With a facing slide, also called boring head, you can perform almost all turning operations with fewer different tools. The slide position of the facing slide in the X direction can be programmed. On the facing slide you mount, for example, a longitudinal turning tool that you call with a TOOL CALL block.

Machining also works with a tilted working plane and on workpieces that are not rotationally symmetric.

Please note while programming

The following constraints apply to the use of a facing slide:

- Miscellaneous functions **M91** and **M92** cannot be used
- Retraction with **M140** is not possible
- **TCPM** or **M128** are not possible
- **DCM** collision monitoring cannot be used
- Cycles 800, 801 and 880 cannot be used

If you are using the facing slide in the tilted working plane, please note the following:

- The control calculates the tilted working plane as in milling mode. The **COORD ROT** and **TABLE ROT** functions, as well as **SYM (SEQ)**, refer to the XY plane.
- HEIDENHAIN recommends using the **TURN** positioning behavior. The **MOVE** positioning behavior is not the best option in combination with the facing slide.

NOTICE

Caution: Danger to the tool and workpiece!

For the deployment of a facing slide, a kinematic model prepared by the machine manufacturer must be selected by means of the function **FUNCTION MODE TURN**. In this kinematic model, the control implements the programmed X-axis movements of the facing slide as U-axis movements when the **FACING HEAD** function is active. This automatism does not work if the **FACING HEAD** function is inactive and in the **Manual operation** operating mode, which means that **X** movements (programmed or axis key) are executed in the X axis. In this case, the facing slide must be moved with the U axis. There is a risk of collision during retraction or manual movements!

- Position facing slide at home position with active **FACING HEAD POS** function
- Retract facing slide with active **FACING HEAD POS** function
- In **Manual operation** mode, move the facing slide with the **U** axis key
- Because the **Tilt the working plane** function is possible, pay attention to the 3-D ROT status

Entering tool data

The tool data correspond to the data from the turning-tool table.

Further information: User's Manual for Setup, Testing and Running NC Programs

Please note for tool calls:

- **TOOL CALL** block without tool axis
- Cutting speed and spindle speed with **TURNDATA SPIN**
- Switch the spindle on with **M3** or **M4**

To set a spindle speed limitation you can use the **NMAX** value from the tool table as well as **SMAX** value from **FUNCTION TURNDATA SPIN**.

Activating and positioning the facing slide function

Before you can activate the facing slide function, you have to select a kinematic model with facing slide by means of **FUNCTION MODE TURN**. The machine tool builder provides this kinematic model.

Example

5 FUNCTION MODE TURN "FACINGHEAD"	Switchover to turning mode with facing slide

> Upon activation, the facing slide automatically moves to the datum in the X and Y axes. Position the spindle axis to clearance height beforehand or enter the clearance height in the **FACING HEAD POS** NC block.

Activate the facing slide function as follows:

SPEC FCT
- Press the **SPEC FCT** key

TURNING PROGRAM FUNCTIONS
- Press the **TURNING PROGRAM FUNCTIONS** soft key

FACING SLIDE
- Press the **FACING SLIDE** soft key

FACING HEAD POS
- Press the **FACING HEAD POS** soft key
- Enter the clearance height, if required
- Enter enter the feed rate, if required

Example

7 FACING HEAD POS	Activating without clearance height
7 FACING HEAD POS HEIGHT+100 FMAX	Activating with positioning to clearance height Z+100 at rapid traverse

Working with the facing slide

⚙	Refer to your machine manual! The machine tool builder can provide his own cycles for working with a facing slide. The standard functional range is described below.

You machine tool builder can provide a feature with which you can specify the position with an offset of the facing slide in X direction. The datum always has to be in the spindle axis, however.

Recommended program structure:

1. Activate **FUNCTION MODE TURN** with facing slide
2. Move to safe position, if necessary
3. Shift the datum to the spindle axis
4. Activate and position the facing slide with **FACING HEAD POS**
5. Perform machining in ZX coordinate plane using turning cycles
6. Retract facing slide and move to home position
7. Deactivate facing slide
8. Switch over machining mode with **FUNCTION MODE TURN** or **FUNCTION MODE MILL**

The coordinate plane is defined such that the X coordinates describe the diameter of the workpiece and the Z coordinates the longitudinal positions.

Deactivating the facing slide function

Deactivate the facing slide function as follows:

SPEC FCT
- Press the **SPEC FCT** key

TURNING PROGRAM FUNCTIONS
- Press the **TURNING PROGRAM FUNCTIONS** soft key

FACING SLIDE
- Press the **FACING SLIDE** soft key

FUNCTION FACING HEAD
-

ENT
- Press the **ENT** key

Example

7 FUNCTION FACING HEAD OFF	Deactivating the facing slide

Cutting force monitoring with the AFC function

Refer to your machine manual!

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

You can also use the **AFC** function (option 45) in turning mode and thus monitor the complete machining process. In turning mode, the control checks for tool wear and tool breakage.

For this purpose, the control uses the reference load **Pref**, the minimum load **Pmin** and the maximum load **Pmax**.

Cutting force monitoring with **AFC** basically works like adaptive feed control in milling mode. The control requires slightly different data, which you provide via the table AFC.TAB.

Execute the function **AFC CUT BEGIN** only after the starting rotational speed has been reached. If this is not the case, then the control issues an error message, and the AFC cut is not started.

Further information: User's Manual for Setup, Testing and Running NC Programs

Defining the AFC basic settings

The table AFC.TAB is valid for milling and turning mode. For turning mode, you define your own monitoring settings (line in the table).

Enter the following data in the table:

Column	Function
NR	Consecutive line number in table
AFC	Name of the monitoring setting. You enter this name in the **AFC** column of the tool table. It specifies the assignment to the tool.
FMIN	Feed rate at which the control is to perform an overload reaction. Input value in turning mode: 0 (not required in turning mode)
FMAX	Maximum feed rate in the material up to which the control can automatically increase the feed rate. Input value in turning mode: 0 (not required in turning mode)
FIDL	Feed rate for traverse when the tool is not cutting (feed rate in the air). Input value in turning mode: 0 (not required in turning mode)
FENT	Feed rate at which the control is to traverse when the tool enters or exits the material. Input value in turning mode: 0 (not required in turning mode)
OVLD	Desired reaction of the control to overload: ■ **S** / **E** / **F**: Display error message on the screen ■ **L**: Disable active tool ■ **-**: No overload reaction In turning mode it is not possible to insert replacement tools. If you define the overload reaction **M**, the control outputs an error message.
POUT	Entering the minimum load **Pmin** for tool breakage monitoring
SENS	Sensitivity of the feed control Input value in turning mode: 0 or 1 ■ SENS 1: Pmin is evaluated ■ SENS 0: Pmin is not evaluated
PLC	Value that the control is to transfer to the PLC at the beginning of a machining step. The machine tool builder defines the function, so refer to your machine manual.

Defining the monitoring setting for turning tools

Enter a separate monitoring setting for each turning tool. Proceed as follows:

- To open the tool table TOOL.T
- Find turning tool
- Enter the appropriate setting in the AFC column

If you are using with the extended tool management, you can also enter the monitoring settings directly in the Tool form.

Performing a teach-in cut

In turning mode, the teach-in phase has to be run completely. The control generates an error message if you enter **TIME** or **DIST** for the **AFC CUT BEGIN** function.

Canceling with the **EXIT LEARNING** softy key is not permitted.

You cannot reset the reference load, the **PREF RESET** soft key is dimmed.

Activating and deactivating AFC

You activate the feed control as in milling mode.

Monitoring tool wear and tool breakage

In turning mode, the control can check for tool wear and tool breakage.

A tool breakage leads to a sudden load decrease. If you want the control to monitor the load decrease, too, enter the value 1 in the SENS column.

Further information: User's Manual for Setup, Testing and Running NC Programs

15

Grinding

15.1 Grinding operations on milling machines (option 156)

Introduction

Refer to your machine manual!
Grinding must be configured and enabled by your machine tool builder. Some functions and cycles may not be available on your machine tool.

Special types of milling machines allow performing both milling and grinding operations. A workpiece can thus be machined completely on one machine, even if complex milling and grinding operations are required.

The term grinding encompasses many types of machining that differ in quite a few respects, e.g.:

- Jig grinding
- Cylindrical grinding
- Surface grinding

The TNC 640 currently features jig grinding.

Grinding tools

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

If grinding is enabled on your milling machine, (Option 156), the dressing function is also available. This means that you can shape or resharpen the grinding wheel in the machine.

Further information: User's Manual for Setup, Testing and Running NC Programs

Jig grinding

> The control provides various cycles for the specific jig grinding or dressing movements.
> **Further information:** Cycle Programming User's Manual

Jig grinding means grinding of a 2-D contour. The tool movement in the plane may be superimposed by a reciprocation movement along the active tool axis.

On a milling machine, jig grinding will mainly be used for finishing a pre-machined contour with a grinding tool. There is not much of a difference between jig grinding and milling. Instead of a milling cutter, a grinding tool is used, such as a grinding pin or a grinding wheel. Jig grinding produces more precise results and a better surface quality than milling.

Machining is performed in milling mode, i.e. with **FUNCTION MODE MILL**.

Grinding cycles provide special movements for the grinding tool. A stroke or oscillating movement, the so-called reciprocating stroke, is superimposed with the movement in the working plane.

Grinding is also possible with a tilted working plane. The tool reciprocates along the active tool axis in the current working plane (WPL-CS).

Reciprocating stroke

For jig grinding, the movement of the tool in the plane can be superimposed with a stroke movement, the so-called reciprocating stroke. The superimposed stroke movement is effective in the active tool axis.

You define an upper and a lower stroke limit and can start and stop the reciprocating stroke and reset the corresponding values. The reciprocating stroke is effective until you stop it. **M30** will stop the reciprocating stroke automatically.

The control provides cycles for defining, starting, and stopping reciprocating strokes.

As long as the reciprocating stroke is active in a started NC program, you cannot change to the **Manual Operation** or **Positioning w/ Manual Data Input** operating modes.

> The reciprocating stroke continues to be effective during a programmed stop with **M0** and after the end of an NC block in the **Program run, single block** operating mode.

> The control does not support block scans while the reciprocating stroke is active.

Graphic display of the reciprocating stroke

The graphic simulation in the **Program run, single block** and **Program run, full sequence** operating modes shows the superimposed stroke movement.

Structure of the NC program

An NC program for grinding is structured as follows:

- Dressing of the grinding tool, if required
- Defining the reciprocating stroke
- If necessary, explicitly starting the reciprocating stroke
- Moving along the contour
- Stopping the reciprocating stroke

For the contour, you can select specific machining cycles, such as pocket, stud, or SL cycles.

With a grinding tool, the control behaves in the same way as with a milling cutter.

- If no cycle is programmed and you move along a contour whose smallest inside radius is smaller than the tool radius, the control will display an error message.
- If you machine with SL cycles, only those areas will be ground that are suitable for the given tool radius. In this case, the resulting contour will not be completely finished and may need to be reworked.

Further information: Cycle Programming User's Manual

Compensation in the grinding process

In order to achieve the required precision, you can use compensation tables during jig grinding.

Further information: "Compensation table", Page 377

15.2 Dressing (option 156)

Dressing function fundamentals

Refer to your machine manual!

For dressing operations, the machine must be prepared accordingly by the machine manufacturer. The machine manufacturer may provide his own cycles.

The term “dressing” refers to the sharpening or trueing up of a grinding tool inside the machine. During dressing, the dresser machines the grinding wheel. Thus, in dressing, the grinding tool is the workpiece.

The dresser removes material and thereby changes the dimensions of the grinding wheel. Dressing the diameter, for example, causes the radius of the grinding wheel to become smaller.

Not all grinding tools require dressing. Comply with the information provided by your tool manufacturer.

Coordinate planes for dressing

In dressing, the workpiece datum is located on an edge of the grinding wheel. Select the respective edge using Cycle 1030 **ACTIVATE WHEEL EDGE**.

During dressing, the axes are arrayed such that the X coordinates describe positions on the radius of the grinding wheel, and the Z coordinates describe the positions along the axis of the grinding wheel. The dressing programs are thus not contingent on the machine type.

The machine manufacturer defines which machine axes will perform the programmed movements.

Simplified dressing

Refer to your machine manual!

For dressing operations, the machine must be prepared accordingly by the machine manufacturer. The machine manufacturer may provide his own cycles.

Your machine manufacturer can program the entire dressing mode in a macro. Depending on this macro, you can start dressing mode either with Cycle 1010 **DRESSING DIAMETER**, with Cycle 1015 **PROFILE DRESSING**, or with an OEM cycle.

It is not necessary to program **FUNCTION DRESS BEGIN**.

In this case, the machine manufacturer determines the dressing sequence.

Programming with FUNCTION DRESS

Refer to your machine manual!

Dressing mode is a machine-dependent function. Your machine manufacturer may provide you with a simplified procedure.

Further information: "Simplified dressing", Page 541

NOTICE

Danger of collision!

When you activate **FUNCTION DRESS BEGIN**, the kinematics is switched over. The grinding wheel becomes the workpiece. The axes may move in the opposite direction. There is a risk of collision during the execution of the function and during the subsequent machining!

- Before starting **FUNCTION DRESS BEGIN**, position the grinding wheel near the dressing tool
- Activate the **FUNCTION DRESS**dressing mode in the **Program run, single block** or **Program run, full sequence** operating mode only
- Once you have activated **FUNCTION DRESS BEGIN**, use exclusively cycles from HEIDENHAIN or from your machine tool builder

NOTICE

Danger of collision!

The dressing cycles position the dressing tool at the programmed grinding wheel edge. Positioning is done in three axes at the same time. The control does not perform collision checking during this movement!

- Before starting **FUNCTION DRESS BEGIN**, position the grinding wheel near the dressing tool
- Make sure there is no risk of collision
- Verify the NC program by slowly executing it block by block

Operating notes

- The grinding tool must not be assigned a too carrier kinematic model.
- The control does not graphically depict the dressing operation. The times determined by the simulation do not reflect the actual machining times. One reason for this is the necessary switching of the kinematic model.
- With the switch to dressing mode, the grinding tool remains in the spindle and retains its current rotational speed.

The control does not support a block scan during the dressing process. If, during a block scan, you select the first NC block after the dressing operation, then the control moves to the most recently approached position in the dressing operation.

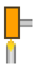
Programming notes

- The **FUNCTION DRESS BEGIN** function is allowed only if a grinding tool is in the spindle.
- If the "tilt working plane" function or **TCPM** function is active, then you cannot switch to dressing mode.
- No coordinate transformation cycles are permitted in dressing mode.
- The **M140** function is not allowed in dressing mode.
- During dressing, the cutting edge of the dresser must be at the same height as the grinding wheel. The programmed Y coordinate must be 0.

Switching between normal operation and dressing mode

For the control to switch to the kinematic model for dressing, you must program the dressing process between the functions **FUNCTION DRESS BEGIN** and **FUNCTION DRESS END**.

If the dressing mode is active, then the control shows a symbol in the status display.

Icon	Mode
	Dressing mode active: **FUNCTION DRESS BEGIN**
No icon	Normal milling or jig grinding operation is active

You can switch back to normal operation with the function **FUNCTION DRESS END**.

In the event of an NC program abort or a power interruption, the control automatically activates normal operation and the kinematic model that was active prior to dressing mode.

NOTICE

Danger of collision!

With an active kinematic model, the machine movements may be in the opposite direction. There is a risk of collision when you move the axes!

- After an NC program abort or power interruption, check the direction of traverse of the axes
- Program a switch in the kinematic model as needed

Activating dressing mode

To activate dressing mode, proceed as follows:

SPEC FCT

- Press the **SPEC FCT** key

PROGRAM FUNCTIONS

- Press the **PROGRAM FUNCTIONS** soft key

FUNCTION DRESS

- Press the **FUNCTION DRESS** soft key

FUNCTION DRESS BEGIN

- Press the **FUNCTION DRESS BEGIN** soft key

If the machine manufacturer has enabled the selection of kinematic models, then proceed as follows:

SELECT

- Press the **SELECT KINEMATICS** soft key
- Pre-position the dresser and the center of the grinding tool in the Y coordinate in proper relation to each other

Example

11 FUNCTION DRESS BEGIN	Activate dressing mode
12 FUNCTION DRESS BEGIN "KINE_DRESS"	Activate dressing mode with kinematic model selection

You can switch back to normal operation with the function **FUNCTION DRESS END**.

Example

18 FUNCTION DRESS END	Deactivate dressing mode

16

Operating the touchscreen

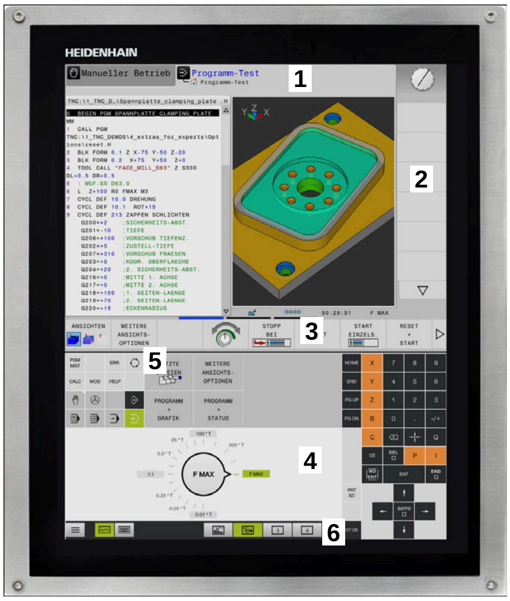
16.1 Display unit and operation

Touchscreen

> Refer to your machine manual!
> This function must be enabled and adapted by the machine tool builder.

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

As an alternative, the TNC 640 has its operating panel integrated in the 19" screen.



1 Header
When the control is on, the screen displays the selected operating modes in the header.

2 Soft-key row for the machine tool builder

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

4 Integrated operating panel

5 Setting the screen layout

6 Switchover between machine operating modes, programming modes, and a third desktop

Operating panel

Depending on the version, the control can still be operated through the operating panel. Touch operation with gestures works as well.

If you have a control with integrated operating panel, the following description applies:

Integrated operating panel

The operating panel is integrated in the screen. The content of the operating panel changes depending on the current operating mode.

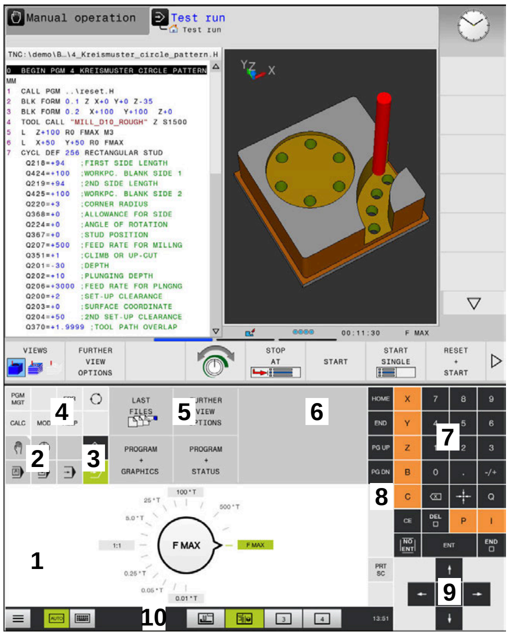
1 Area for showing the following:
   - Alphabetic keyboard
   - HEROS menu
   - Potentiometer for the speed of simulation (only in the **Test Run** operating mode)
2 Machine operating modes
3 Programming modes

   The control shows the active operating mode, to which the screen is switched, with a green background.

   The control shows the operating mode in the background through a small white triangle.
4
   - File management
   - Calculator
   - MOD function
   - HELP function
   - Show error messages
5 Rapid access menu

   Depending on the operating mode, you'll find the most important functions here at a glance.
6 Opening the programming dialogs (only in the **Programming** and **Positioning w/ Manual Data Input** operating modes)
7 Numerical input and axis selection
8 Navigation
9 Arrows and the jump statement **GOTO**
10 Task bar

   Further information: User's Manual for Setup, Testing and Running NC Programs



Operating panel of the Test Run mode

Operating panel in the Manual Operation mode

In addition, the machine tool builder supplies a machine operating panel.

> Refer to your machine manual!
>
> External keys, e.g.**NC START** or **NC STOP**, are described in your machine manual.

Basic operation

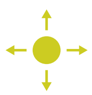
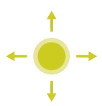
The following keys, for example, can easily be replaced by hand gestures:

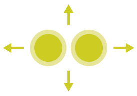
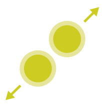
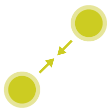
Key	Function	Gesture
	Switch between operating modes	Tap on the operating mode in the header
	Shift the soft-key row	Swipe horizontally over the soft-key row
	Soft-key selection keys	Tap on the function in the touchscreen

16.2 Gestures

Overview of possible gestures

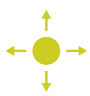
The screen of the control is multi-touch capable. That means that it can distinguish various gestures, including with two or more fingers at once.

Symbol	Gesture	Meaning
	Tap	A brief touch by a finger on the screen
	Double tap	Two brief touches on the screen
	Long press	Continuous contact of fingertip on the screen
	Swipe	Flowing motion over the screen
	Drag	A combination of long-press and then swipe, moving a finger over the screen when the starting point is clearly defined

Symbol	Gesture	Meaning
	Two-finger drag	A combination of long-press and then swipe, moving two fingers in parallel over the screen when the starting point is clearly defined
	Spread	Two fingers long-press and move away from each other
	Pinch	Two fingers move toward each other

Navigating in the table and NC programs

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

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

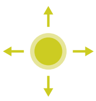
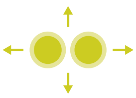
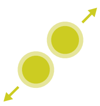
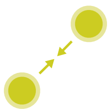
Operating the simulation

The control offers touch operation with the following graphics:

- Programming graphics in the **Programming** operating mode
- 3-D view in the **Test Run** operating mode
- 3-D view in the **Program Run Single Block** operating mode
- 3-D view in the **Program Run Full Sequence** operating mode
- Kinematics view

Rotate, zoom or move a graphic

The control supports the following gestures:

Symbol	Gesture	Function
	Double tap	Set the graphic to its original size
	Drag	Rotate the graphic (only 3-D graphics)
	Two-finger drag	Move graphics
	Spread	Magnify the graphic
	Pinch	Reduce the graphic

Measure the graphic

If you have activated measurement in the **Test Run** operating mode, you have the following additional function:

Symbol	Gesture	Function
	Tap	Select the measuring point

Operating the CAD viewer

The control also supports touch operation for working with the **CAD-Viewer**. You have various gestures available depending on the operating mode.

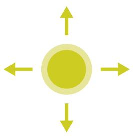
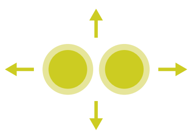
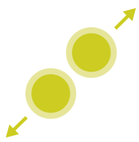
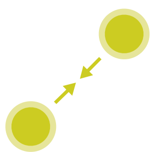
To be able to use all applications, first use the icon to select the desired function:

Icon	Function
	Default setting
	**Add** Works in the selection mode like a pressed **Shift** key
	Remove Works in the selection mode like a pressed **CTRL** key

Layer setting mode and specify the workpiece preset

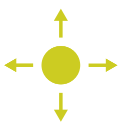
The control supports the following gestures:

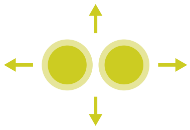
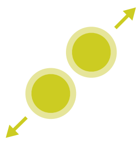
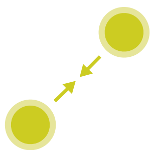
Symbol	Gesture	Function
	Tap on an element	Show element information Specify the workpiece preset
	Double-tap on the background	Set the graphic or 3-D model to its original size

Symbol	Gesture	Function
	Activate **Add** and double-tap on the background	Reset the graphic or 3-D model to its original size and angle
	Drag	Rotate the graphic or 3-D model (only in the Layer Setting mode)
	Two-finger drag	Move a graphic or 3-D model
	Spread	Enlarge a graphic or 3-D model
	Pinch	Reduce a graphic or 3-D model

Selecting a contour

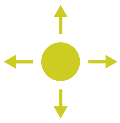
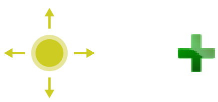
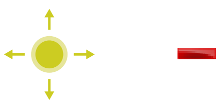
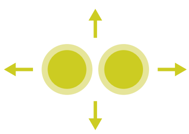
The control supports the following gestures:

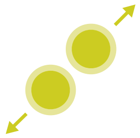
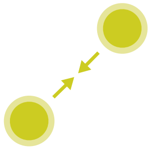
Symbol	Gesture	Function
	Tap on an element	Select element
	Tap on an element in the list-view window	Select or deselect an element
	Activate **Add** and tap on an element	Part, shorten, or lengthen and element
	Activate **Remove** and tap on an element	Deselect an element
	Double-tap on the background	Reset the graphic to its original size
	Swipe over an element	Show a preview of selected elements Show element information

Symbol	Gesture	Function
	Two-finger drag	Move graphics
	Spread	Magnify the graphic
	Pinch	Reduce the graphic

Selecting machining positions

The control supports the following gestures:

Symbol	Gesture	Function
	Tap on an element	Select element Selecting an intersection
	Double-tap on the background	Reset the graphic to its original size
	Swipe over an element	Show a preview of selected elements Show element information
	Activate **Add** and drag	Spread a fast selection area
	Activate **Remove** and drag	Spread an area for deselection of elements
	Two-finger drag	Move graphics

Symbol	Gesture	Function
	Spread	Magnify the graphic
	Pinch	Reduce the graphic

Save elements and switch to the NC program

When you tap on the appropriate icons, the controls saves the selected elements.

You can switch back to the **Programming** operating mode in the following ways:

- Press the **Programming** key

  The control switches to the **Programming** mode of operation.
- Close the **CAD-Viewer**

  The control automatically switches to the **Programming** operating mode.
- Use the task bar to leave the **CAD-Viewer** open on the third desktop

  The third desktop stays active in the background

17

Tables and overviews

17.1 System data

List of FN 18 functions

With the **FN 18: SYSREAD** function, you can read system data and save them to Q parameters. The selection of the system datum occurs via a group number (ID no.), a system data number, and, if necessary, an index.

> The read values of the function **FN 18: SYSREAD** are always output by the control in **metric** units regardless of the NC program's unit of measure.

The following is a complete list of the **FN 18: SYSREAD** function. Please be aware that not all functions are available depending on the model of your control.

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Program information				
	10	3	-	Number of the active machining cycle
		6	-	Number of the most recently executed touch probe cycle –1 = None
		7	-	Type of calling NC program: –1 = None 0 = Visible NC program 1 = Cycle/macro, main program is visible 2 = Cycle/macro, there is no visible main program
		103	Q parameter number	Relevant within NC cycles; for inquiry as to whether the Q parameter given under IDX was explicitly stated in the associated CYCLE DEF.
		110	QS parameter number	Is there a file with the name QS(IDX)? 0 = No, 1 = Yes This function eliminates relative file paths.
		111	QS parameter number	Is there a directory with the name QS(IDX)? 0 = no, 1 = Yes Only absolute directory paths are possible.

Group name	Group number ID...	System data number NO....	Index IDX...	Description
System jump addresses				
	13	1	-	Label number or label name (string or QS) jumped to during M2/M30 instead of ending the current NC program. Value = 0: M2/M30 have the normal effect
		2	-	Label number or label name (string or QS) jumped to in the event of FN14: ERROR with the NC CANCEL reaction instead of aborting the NC program with an error message. The error number programmed in the FN14 command can be read under ID992 NR14. Value = 0: FN14 has the normal effect.
		3	-	Label number or label name (string or QS) jumped to in the event of an internal server error (SQL, PLC, CFG) or with erroneous file operations (FUNCTION FILECOPY, FUNCTION FILEMOVE, or FUNCTION FILEDELETE) instead of aborting the NC program with an error message. Value = 0: Error has the normal effect.
Indexed access to Q parameters				
	15	10	Q parameter number	Reads Q(IDX)
		11	QL parameter no.	Reads QL(IDX)
		12	QR parameter no.	Reads QR(IDX)
Machine status				
	20	1	-	Active tool number
		2	-	Prepared tool number
		3	-	Active tool axis 0 = X 6 = U 1 = Y 7 = V 2 = Z 8 = W
		4	-	Programmed spindle speed
		5	-	Active spindle condition -1 = spindle condition not defined 0 = M3 active 1 = M4 active 2 = M5 active after M3 3 = M5 active after M4
		7	-	Active gear range
		8	-	Active coolant status 0 = off, 1 = on
		9	-	Active feed rate
		10	-	Index of prepared tool
		11	-	Index of active tool

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		14	-	Number of active spindle
		20	-	Programmed cutting speed in turning operation
		21	-	Spindle mode in turning mode: 0 = constant speed 1 = constant cutting speed
		22	-	Coolant status M7: 0 = inactive, 1 = active
		23	-	Coolant status M8: 0 = inactive, 1 = active
Channel data				
	25	1	-	Channel number
Cycle parameters				
	30	1	-	Set-up clearance
		2	-	Hole depth / milling depth
		3	-	Plunging depth
		4	-	Feed rate for plunging
		5	-	First side length of pocket
		6	-	Second side length of pocket
		7	-	First side length of slot
		8	-	Second side length of slot
		9	-	Radius of circular pocket
		10	-	Feed rate for milling
		11	-	Rotational direction of the milling path
		12	-	Dwell time
		13	-	Thread pitch for Cycles 17 and 18
		14	-	Finishing allowance
		15	-	Roughing angle
		21	-	Probing angle
		22	-	Probing path
		23	-	Probing feed rate
		49	-	HSC mode (Cycle 32 Tolerance)
		50	-	Tolerance for rotary axes (Cycle 32 Tolerance)
		52	Q parameter number	Type of transfer parameter for user cycles: –1: Cycle parameter not programmed in CYCL DEF 0: Cycle parameter numerically programmed in CYCL DEF (Q parameter) 1: Cycle parameter programmed as string in CYCL DEF (Q parameter)
		60	-	Clearance height (touch probe cycles 30 to 33)

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		61	-	Inspection (touch probe cycles 30 to 33)
		62	-	Cutting edge measurement (touch probe cycles 30 to 33)
		63	-	Q parameter number for the result (touch probe cycles 30 to 33)
		64	-	Q parameter type for the result (touch probe cycles 30 to 33) 1 = Q, 2 = QL, 3 = QR
		70	-	Multiplier for feed rate (cycles 17 and 18)
Modal status				
	35	1	-	Dimensions: 0 = absolute (G90) 1 = incremental (G91)
		2	-	Radius compensation: 0 = R0 1 = RR/RL 10 = Face milling 11 = Peripheral milling
Data for SQL tables				
	40	1	-	Result code for the last SQL command. If the last result code was 1 (=error), the error code is transferred as the return code.
Data from the tool table				
	50	1	Tool no.	Tool length L
		2	Tool no.	Tool radius R
		3	Tool no.	Tool radius R2
		4	Tool no.	Oversize for tool length DL
		5	Tool no.	Tool radius oversize DR
		6	Tool no.	Tool radius oversize DR2
		7	Tool no.	Tool locked TL 0 = not locked, 1 = locked
		8	Tool no.	Number of the replacement tool RT
		9	Tool no.	Maximum tool age TIME1
		10	Tool no.	Maximum tool age TIME2
		11	Tool no.	Current tool age CUR.TIME
		12	Tool no.	PLC status
		13	Tool no.	Maximum tooth length LCUTS
		14	Tool no.	Maximum plunge angle ANGLE
		15	Tool no.	TT: Number of tool teeth CUT
		16	Tool no.	TT: Wear tolerance for length, LTOL
		17	Tool no.	TT: Wear tolerance for radius, RTOL

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		18	Tool no.	TT: Direction of rotation DIRECT 0 = positive, –1 = negative
		19	Tool no.	TT: Offset in plane R-OFFS R = 99999.9999
		20	Tool no.	TT: Offset in length L-OFFS
		21	Tool no.	TT: Breakage tolerance for length, LBREAK
		22	Tool no.	TT: Breakage tolerance for radius, RBREAK
		28	Tool no.	Maximum speed NMAX
		32	Tool no.	Point angle TANGLE
		34	Tool no.	LIFTOFF allowed (0 = No, 1 = Yes)
		35	Tool no.	Wear tolerance for radius R2TOL
		36	Tool no.	Tool type TYPE (miller = 0, grinder = 1, ... touch probe = 21)
		37	Tool no.	Corresponding line in the touch-probe table
		38	Tool no.	Timestamp of last use
		39	Tool no.	ACC
		40	Tool no.	Pitch for thread cycles
		41	Tool no.	AFC: reference load
		42	Tool no.	AFC: overload early warning
		43	Tool no.	AFC: overload NC stop

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Data from the pocket table				
	51	1	Pocket number	Tool number
		2	Pocket number	0 = no special tool 1 = special tool
		3	Pocket number	0 = no fixed pocket 1 = fixed pocket
		4	Pocket number	0 = pocket not locked 1 = pocket locked
		5	Pocket number	PLC status
Determine the tool pocket				
	52	1	Tool no.	Pocket number
		2	Tool no.	Tool magazine number
Tool data for T and S strobes				
	57	1	T code	Tool number IDX0 = T0 strobe (store tool), IDX1 = T1 strobe (load tool), IDX2 = T2 strobe (prepare tool)
		2	T code	Tool index IDX0 = T0 strobe (store tool), IDX1 = T1 strobe (load tool), IDX2 = T2 strobe (prepare tool)
		5	-	Spindle speed IDX0 = T0 strobe (store tool), IDX1 = T1 strobe (load tool), IDX2 = T2 strobe (prepare tool)
Values programmed in TOOL CALL				
	60	1	-	Tool number T
		2	-	Active tool axis 0 = X 1 = Y 2 = Z 6 = U 7 = V 8 = W
		3	-	Spindle speed S
		4	-	Oversize for tool length DL
		5	-	Tool radius oversize DR
		6	-	Automatic TOOL CALL 0 = Yes, 1 = No
		7	-	Tool radius oversize DR2
		8	-	Tool index
		9	-	Active feed rate
		10	-	Cutting speed [mm/min]

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Values programmed in TOOL DEF				
	61	0	Tool no.	Read the number of the tool change sequence: 0 = Tool already in spindle, 1 = Change between external tools, 2 = Change from internal to external tool, 3 = Change from special tool to external tool, 4 = Load external tool, 5 = Change from external to internal tool, 6 = Change from internal to internal tool, 7 = Change from special tool to internal tool, 8 = Load internal tool, 9 = Change from external tool to special tool, 10 = Change from special tool to internal tool, 11 = Change from special tool to special tool, 12 = Load special tool, 13 = Unload external tool, 14 = Unload internal tool, 15 = Unload special tool
		1	-	Tool number T
		2	-	Length
		3	-	Radius
		4	-	Index
		5	-	Tool data programmed in TOOL DEF 1 = Yes, 0 = No

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Values programmed with FUNCTION TURNDATA				
	62	1	-	Tool length oversize DXL
		2	-	Tool length oversize DYL
		3	-	Tool length oversize DZL
			-	Cutting radius oversize DRS
Values for LAC and VSC				
	71	0	0	Index of the NC axis for which the LAC weighing run will be performed or was last performed (X to W = 1 to 9)
			2	Total inertia determined by the LAC weighing run in [kgm²] (with A/B/C rotary axes) or total mass in [kg] (with X/Y/Z linear axes)
		1	0	Cycle 957 Retraction from thread
		2	0	Number of the last VSC cycle that was called
Freely available memory area for OEM cycles				
	72	0-39	0 to 30	Freely available memory area for OEM cycles. The values are only reset by the control during a control reboot (= 0). With "Cancel," the values are not reset to the value that they had at the time of execution. Up to and including 597110-11: only NR 0-9 and IDX 0-9 Starting with 597110-12: NR 0-39 and IDX 0-30
Freely available memory area for user cycles				
	73	0-39	0 to 30	Freely available memory area for user cycles The values are only reset by the control during a control reboot (= 0). With "Cancel," the values are not reset to the value that they had at the time of execution. Up to and including 597110-11: only NR 0-9 and IDX 0-9 Starting with 597110-12: NR 0-39 and IDX 0-30
Read minimum and maximum spindle speed				
	90	1	Spindle ID	Minimum spindle speed of the lowest gear range. If no gear stages are configured, CfgFeedLimits/minFeed of the first parameter set of the spindle is evaluated. Index 99 = active spindle
		2	Spindle ID	Maximum spindle speed from the highest gear stage. If no gear stages are configured, CfgFeedLimits/maxFeed of the first parameter set of the spindle is evaluated. Index 99 = active spindle

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Tool compensation				
	200	1	1 = without oversize 2 = with oversize 3 = with oversize and oversize from TOOL CALL	Active radius
		2	1 = without oversize 2 = with oversize 3 = with oversize and oversize from TOOL CALL	Active length
		3	1 = without oversize 2 = with oversize 3 = with oversize and oversize from TOOL CALL	Rounding radius R2
		6	Tool no.	Tool length Index 0= active tool
Coordinate transformations				
	210	1	-	Basic rotation (manual)
		2	-	Programmed rotation
		3	-	Active mirror axis. Bits 0 to 2 and 6 to 8: Axes X, Y, Z and U, V, W
		4	Axis	Active scaling factor Index: 1 - 9 (X, Y, Z, A, B, C, U, V, W)
		5	Rotary axis	3D-ROT Index: 1 - 3 (A, B, C)
		6	-	Tilt working plane in Program Run operating modes 0 = Not active –1 = Active
		7	-	Tilt working plane in Manual operating modes 0 = Not active –1 = Active
		8	QL parameter no.	Angle of misalignment between spindle and tilted coordinate system. Projects the angle specified in the QL parameter from the input coordinate system to the tool coordinate system. If IDX is omitted, the angle 0 is used for projection.

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		10	-	Type of definition of the active tilt: 0 = no tilt—is returned if, both in **Manual Operation** and in the automatic modes, no tilt is active. 1 = axial 2 = spatial angle

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Active coordinate system				
	211	–	-	1 = input system (default) 2 = REF system 3 = tool change system
Special transformations in turning mode				
	215	1	-	Angle for the precession of the input system in the XY plane in turning mode To reset the transformation the value 0 must be entered for the angle. This transformation is used in connection with Cycle 800 (parameter Q497).
		3	1-3	Reading out of the spatial angle written with NR2 Index: 1 - 3 (redA, redB, redC)
Current datum shift				
	220	2	Axis	Current datum shift in [mm] Index: 1 - 9 (X, Y, Z, A, B, C, U, V, W)
		3	Axis	Read the difference between reference point and preset. Index: 1 - 9 (X, Y, Z, A, B, C, U, V, W)
		4	Axis	Read values for OEM offset.. Index: 1 - 9 (X_OFFS, Y_OFFS, Z_OFFS,...)
Traverse range				
	230	2	Axis	Negative software limit switches Index: 1 - 9 (X, Y, Z, A, B, C, U, V, W)
		3	Axis	Positive software limit switches Index: 1 - 9 (X, Y, Z, A, B, C, U, V, W)
		5	-	Software limit switch on or off: 0 = on, 1 = off For modulo axes, either both the upper and lower limits or no limit at all must be set.
Read the nominal position in the REF system				
	240	1	Axis	Current nominal position in the REF system
Read the nominal position in the REF system, including offsets (handwheel, etc.)				
	241	1	Axis	Current nominal position in the REF system
Read the current position in the active coordinate system				
	270	1	Axis	Current nominal position in the input system When called while tool radius compensation is active, the function supplies the uncompensated positions for the principal axes X, Y, and Z. If the function is called for a rotary axis and tool radius compensation is active, an error message is issued. Index: 1 to 9 (X, Y, Z, A, B, C, U, V, W)
Read the current position in the active coordinate system, including offsets (handwheel, etc.)				
	271	1	Axis	Current nominal position in the input system

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Read information to M128				
	280	1	-	M128 active: –1 = Yes, 0 = No
		3	-	Condition of TCPM after Q No.: Q No. + 0: TCPM active, 0 = no, 1 = yes Q No. + 1: AXIS, 0 = POS, 1 = SPAT Q No. + 2: PATHCTRL, 0 = AXIS, 1 = VECTOR Q No. + 3: Feed rate, 0 = F TCP, 1 = F CONT
Machine kinematics				
	290	5	-	0: Temperature compensation not active 1: Temperature compensation active
		7	-	KinematicsComp: 0: Compensations by KinematicsComp not active 1: Compensations by KinematicsComp active
		10	-	Index of the machine kinematics from Channels/ChannelSettings/CfgKin-List/kinCompositeModels programmed in FUNCTION MODE MILL or FUNCTION MODE TURN –1 = Not programmed.
Read data of the machine kinematics				
	295	1	QS parameter no.	Read the axis names of the active 3-axis kinematics. The axis names are written according to QS(IDX), QS(IDX+1), and QS(IDX+2). 0 = Operation successful
		2	0	Is FACING HEAD POS function active? 1 = Yes, 0 = No
		4	Rotary axis	Read whether the defined rotary axis partici-pates in the kinematic calculation. 1 = Yes, 0 = No (A rotary axis can be excluded from the kinematics calculating using M138.) Index: 4, 5, 6 (A, B, C)
		6	Axis	Angle head: Displacement vector in the basic coordinate system B-CS through angle head Index: 1, 2, 3 (X, Y, Z)
		7	Axis	Angle head: Direction vector of the tool in the basic coordinate system B-CS Index: 1, 2, 3 (X, Y, Z)
		10	Axis	Determine programmable axes. Determine the axis ID associated with the specified axis index (index from CfgAxis/axisList). Index: 1 - 9 (X, Y, Z, A, B, C, U, V, W)

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		11	Axis ID	Determine programmable axes. Determine the index of the axis (X = 1, Y = 2, ...) for the specified axis ID Index: Axis ID (index from CfgAxis/axisList)
Modify the geometrical behavior				
	310	20	Axis	Diameter programming: –1 = on, 0 = off
Current system time				
	320	1	0	System time in seconds that has elapsed since 01.01.1970, 00:00:00 (real time).
			1	System time in seconds that has elapsed since 01.01.1970, 00:00:00 (look-ahead calculation).
		3	-	Read the processing time of the current NC program.
Formatting of system time				
	321	0	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: DD.MM.YYYY hh:mm:ss
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: DD.MM.YYYY hh:mm:ss
		1	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: D.MM.YYYY h:mm:ss
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: D.MM.YYYY h:mm:ss
		2	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: D.MM.YYYY h:mm
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: D.MM.YYYY h:mm
		3	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: D.MM.YY h:mm
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: D.MM.YY h:mm

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		4	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: YYYY-MM-DD hh:mm:ss
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: YYYY-MM-DD hh:mm:ss
		5	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: YYYY-MM-DD hh:mm
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: YYYY-MM-DD hh:mm
		6	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: YYYY-MM-DD h:mm
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: YYYY-MM-DD h:mm
		7	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: YY-MM-DD h:mm
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: YY-MM-DD h:mm
		8	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: DD.MM.YYYY
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: DD.MM.YYYY
		9	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: D.MM.YYYY
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: D.MM.YYYY

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		10	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: D.MM.YY
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: D.MM.YY
		11	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: YYYY-MM-DD
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: YYYY-MM-DD
		12	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: YY-MM-DD
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: YY-MM-DD
		13	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: hh:mm:ss
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: hh:mm:ss
		14	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: h:mm:ss
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: h:mm:ss
		15	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: h:mm
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: h:mm

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Global Program Settings (GPS): Global activation status				
	330	0	-	0 = No GPS setting is active 1 = Any GPS setting is active
Global Program Settings (GPS): Individual activation status				
	331	0	-	0 = No GPS setting is active 1 = Any GPS setting is active
		1	-	GPS: Basic rotation 0 = Off, 1 = On
		3	Axis	GPS: Mirroring 0 = Off, 1 = On Index: 1 - 6 (X, Y, Z, A, B, C)
		4	-	GPS: Shift in the modified workpiece system 0 = Off, 1 = On
		5	-	GPS: Rotation in input system 0 = Off, 1 = On
		6	-	GPS: Feed rate factor 0 = Off, 1 = On
		8	-	GPS: Handwheel superimpositioning 0 = Off, 1 = On
		10	-	GPS: Virtual tool axis VT 0 = Off, 1 = On
		15	-	GPS: Selection of the handwheel coordinate system 0 = Machine coordinate system M-CS 1 = Workpiece coordinate system W-CS 2 = Modified workpiece coordinate system mW-CS 3 = Working plane coordinate system WPL-CS
		16	-	GPS: Shift in the workpiece system 0 = Off, 1 = On
		17	-	GPS: Axis offset 0 = Off, 1 = On

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Global Program Settings (GPS)				
	332	1	-	GPS: Angle of a basic rotation
		3	Axis	GPS: Mirroring 0 = Not mirrored, 1 = Mirrored Index: 1 - 6 (X, Y, Z, A, B, C)
		4	Axis	GPS: Shift in the modified workpiece coordinate system mW-CS Index: 1 - 6 (X, Y, Z, A, B, C)
		5	-	GPS: Angle of rotation in input coordinate system I-CS
		6	-	GPS: Feed rate factor
		8	Axis	GPS: Handwheel superimpositioning Maximum value Index: 1 - 10 (X, Y, Z, A, B, C, U, V, W, VT)
		9	Axis	GPS: Value for handwheel superimpositioning Index: 1 - 10 (X, Y, Z, A, B, C, U, V, W, VT)
		16	Axis	GPS: Shift in the workpiece coordinate system W-CS Index: 1 - 3 (X, Y, Z)
		17	Axis	GPS: Axis offset Index: 4 - 6 (A, B, C)
TS touch trigger probe				
	350	50	1	Touch probe type: 0: TS120, 1: TS220, 2: TS440, 3: TS630, 4: TS632, 5: TS640, 6: TS444, 7: TS740
			2	Line in the touch-probe table
		51	-	Effective length
		52	1	Effective radius of the stylus tip
			2	Rounding radius
		53	1	Center offset (reference axis)
			2	Center offset (minor axis)
		54	-	Spindle-orientation angle in degrees (center offset)
		55	1	Rapid traverse
			2	Measuring feed rate
			3	Feed rate for pre-positioning: FMAX_PROBE or FMAX_MACHINE
		56	1	Maximum measuring range
			2	Set-up clearance
		57	1	Spindle orientation possible 0=No, 1=Yes
			2	Angle of spindle orientation in degrees

Group name	Group number ID...	System data number NO....	Index IDX...	Description
TT tool touch probe for tool measurement				
	350	70	1	TT: Touch probe type
			2	TT: Line in the tool touch probe table
		71	1/2/3	TT: Touch probe center (REF system)
		72	-	TT: Touch probe radius
		75	1	TT: Rapid traverse
			2	TT: Measuring feed rate with stationary spindle
			3	TT: Measuring feed rate with rotating spindle
		76	1	TT: Maximum probing path
			2	TT: Safety clearance for linear measurement
			3	TT: Safety clearance for radius measurement
			4	TT: Distance from the lower edge of the cutter to the upper edge of the stylus
		77	-	TT: Spindle speed
		78	-	TT: Probing direction
		79	-	TT: Activate radio transmission
		80	-	TT: Stop probing movement upon stylus deflection
Preset from touch probe cycle (probing results)				
	360	1	Coordinate	Last preset of a manual touch probe cycle, or last touch point from Cycle 0 (input coordinate system). Compensations: length, radius, and center offset
		2	Axis	Last preset of a manual touch probe cycle, or last touch point from Cycle 0 (machine coordinate system, only axes from the active 3-D kinematics are allowed as index). Compensation: only center offset
		3	Coordinate	Result of measurement in the input system of touch probe Cycles 0 and 1. The measurement result is read out in the form of coordinates. Compensation: only center offset
		4	Coordinate	Last preset of a manual touch probe cycle, or last touch point from Cycle 0 (workpiece coordinate system) The measurement result is read in the form of coordinates. Compensation: only center offset
		5	Axis	Axis values, not compensated
		6	Coordinate / axis	Readout of the measurement results in the form of coordinates / axis values in the input system from probing operations. Compensation: only length
		10	-	Oriented spindle stop

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		11	-	Error status of probing: 0: Probing was successful –1: Touch point not reached –2: Touch probe already deflected at the start of the probing process
Read values from or write values to the active datum table				
	500	Row number	Column	Read values
Read values from or write values to the preset table (basic transformation)				
	507	Row number	1-6	Read values
Read axis offsets from or write axis offsets to the preset table				
	508	Row number	1-9	Read values
Data for pallet machining				
	510	1	-	Active line
		2	-	Current pallet number. Read value of the NAME column of the last PAL-type entry If the column is empty or does not contain a numerical value, a value of –1 is returned.
		3	-	Active row of the pallet table.
		4	-	Last line of the NC program for the current pallet.
		5	Axis	Tool-oriented editing: Clearance height is programmed: 0 = No, 1 = Yes Index: 1 - 9 (X, Y, Z, A, B, C, U, V, W)
		6	Axis	Tool-oriented editing: Clearance height The value is invalid if ID510 NR5 returns the value 0 with the corresponding IDX. Index: 1 - 9 (X, Y, Z, A, B, C, U, V, W)
		10	-	Row number up to which the pallet table is to be searched during block scan.
		20	-	Type of pallet editing? 0 = Workpiece-oriented 1 = Tool oriented
		21	-	Automatic continuation after NC error: 0 = Locked 1 = Active 10 = Abort continuation 11 = Continuation with the rows in the pallet table that would have been executed next if not for the NC error 12 = Continuation with the row in the pallet table in which the NC error arose 13 = Continuation with the next pallet

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Read data from the point table				
	520	Row number	1-3 X/Y/Z	Read value from active point table.
			10	Read value from active point table.
			11	Read value from active point table.
Read or write the active preset				
	530	1	-	Number of the active preset in the active preset table.
Active pallet preset				
	540	1	-	Number of the active pallet preset. Returns the number of the active preset. If no pallet preset is active, the function returns the value –1.
		2	-	Number of the active pallet preset. As with NR1.
Values for the basic transformation of the pallet preset				
	547	row number	Axis	Read values of the basic transformation from the pallet preset table.. Index: 1 to 6 (X, Y, Z, SPA, SPB, SPC)
Axis offsets from the pallet preset table				
	548	Row number	Offset	Read values of the axis offsets from the pallet preset table.. Index: 1 - 9 (X_OFFS, Y_OFFS, Z_OFFS,...)
OEM offset				
	558	Row number	Offset	Read values for OEM offset.. Index: 1 - 9 (X_OFFS, Y_OFFS, Z_OFFS,...)
Read and write the machine status				
	590	2	1-30	Freely available; not deleted during program selection.
		3	1-30	Freely available; not deleted during a power failure (persistent storage).
Read/write look-ahead parameter of a single axis (at machine level)				
	610	1	-	Minimum feed rate (**MP_minPathFeed**) in mm/min
		2	-	Minimum feed rate at corners (**MP_minCornerFeed**) in mm/min
		3	-	Feed-rate limit for high speeds (**MP_maxG1Feed**) in mm/min
		4	-	Max. jerk at low speeds (**MP_maxPathJerk**) in m/s³
		5	-	Max. jerk at high speeds (**MP_maxPathJerkHi**) in m/s³
		6	-	Tolerance at low speeds (**MP_pathTolerance**) in mm

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		7	-	Tolerance at high speeds (**MP_pathToleranceHi**) in mm
		8	-	Max. derivative of jerk (**MP_maxPathYank**) in m/s^4
		9	-	Tolerance factor for curve machining (**MP_curveTolFactor**)
		10	-	Factor for max. permissible jerk at curvature changes (**MP_curveJerkFactor**)
		11	-	Maximum jerk with probing movements (**MP_pathMeasJerk**)
		12	-	Angle tolerance for machining feed rate (**MP_angleTolerance**)
		13	-	Angle tolerance for rapid traverse (**MP_angleToleranceHi**)
		14	-	Max. corner angle for polygons (**MP_maxPolyAngle**)
		18	-	Radial acceleration with machining feed rate (**MP_maxTransAcc**)
		19	-	Radial acceleration with rapid traverse (**MP_maxTransAccHi**)
		20	Index of physical axis	Max. feed rate (**MP_maxFeed**) in mm/min
		21	Index of physical axis	Max. acceleration (**MP_maxAcceleration**) in m/s^2
		22	Index of physical axis	Maximum transition jerk of the axis in rapid traverse (**MP_axTransJerkHi**) in m/s^2
		23	Index of physical axis	Maximum transition jerk of the axis during machining free rate (**MP_axTransJerk**) in m/s^3
		24	Index of physical axis	Acceleration feedforward control (**MP_compAcc**)
		25	Index of physical axis	Axis-specific jerk at low speeds (**MP_axPathJerk**) in m/s^3
		26	Index of physical axis	Axis-specific jerk at high speeds (**MP_axPathJerkHi**) in m/s^3
		27	Index of physical axis	More precise tolerance examination in corners (**MP_reduceCornerFeed**) 0 = deactivated, 1 = activated
		28	Index of physical axis	DCM: Maximum tolerance for linear axes in mm (**MP_maxLinearTolerance**)
		29	Index of physical axis	DCM: Maximum angle tolerance in [°] (**MP_maxAngleTolerance**)
		30	Index of physical axis	Tolerance monitoring for successive threads (**MP_threadTolerance**)

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		31	Index of physical axis	Form (**MP_shape**) of the **axisCutterLoc** filter 0: Off 1: Average 2: Triangle 3: HSC 4: Advanced HSC
		32	Index of physical axis	Frequency **MP_frequency**) of the **axisCutterLoc** filter in Hz
		33	Index of physical axis	Form (**MP_shape**) of the **axisPosition** filter 0: Off 1: Average 2: Triangle 3: HSC 4: Advanced HSC
		34	Index of physical axis	Frequency (**MP_frequency**) of the **axisPosition** filter in Hz
		35	Index of physical axis	Order of the filter for **Manual** operating mode (**MP_manualFilterOrder**)
		36	Index of physical axis	HSC mode (**MP_hscMode**) of the **axisCutterLoc** filter
		37	Index of physical axis	HSC mode (**MP_hscMode**) of the **axisPosition** filter
		38	Index of physical axis	Axis-specific jerk for probing movements (**MP_axMeasJerk**)
		39	Index of physical axis	Weighting of the filter error for calculating filter deviation (**MP_axFilterErrWeight**)
		40	Index of physical axis	Maximum filter length of position filter (**MP_maxHscOrder**)
		41	Index of physical axis	Maximum filter length of CLP filter (**MP_maxHscOrder**)
		42	-	Maximum feed rate of the axis at machining feed rate (**MP_maxWorkFeed**)
		43	-	Maximum path acceleration at machining feed rate (**MP_maxPathAcc**)
		44	-	Maximum path acceleration at rapid traverse (**MP_maxPathAccHi**)
		51	Index of physical axis	Compensation of following error in the jerk phase (**MP_IpcJerkFact**)
		52	Index of physical axis	kv factor of the position controller in 1/s (**MP_kvFactor**)

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Measure the maximum utilization of an axis				
	621	0	Index of physical axis	Conclude measurement of the dynamic load and save the result in the specified Q parameter.
Read SIK contents				
	630	0	Option no.	You can explicitly determine whether the SIK option given under **IDX** has been set or not. 1 = option is enabled 0 = option is not enabled
		1	-	You can determine whether a Feature Content Level (for upgrade functions) is set, and which one. –1 = No FCL is set <No.> = FCL that is set
		2	-	Read serial number of the SIK -1 = No valid SIK in the system
		10	-	Define the type of control: 0 = iTNC 530 1 = NCK-based control (TNC 640, TNC 620, TNC 320, TNC 128, PNC 610, ...)
Write data for unbalance monitoring				
	850	10	-	Activate and deactivate unbalance monitoring 0 = unbalance monitoring not active 1 = unbalance monitoring active
Counter				
	920	1	-	Planned workpieces. In **Test Run** operating mode the counter generally generates the value 0.
		2	-	Already machined workpieces. In **Test Run** operating mode the counter generally generates the value 0.
		12	-	Workpieces still to be machined. In **Test Run** operating mode the counter generally generates the value 0.
Read and write data of current tool				
	950	1	-	Tool length L
		2	-	Tool radius R
		3	-	Tool radius R2
		4	-	Oversize for tool length DL
		5	-	Tool radius oversize DR
		6	-	Tool radius oversize DR2
		7	-	Tool locked TL 0 = not locked, 1 = locked
		8	-	Number of the replacement tool RT
		9	-	Maximum tool age TIME1

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		10	-	Maximum tool age TIME2 at TOOL CALL
		11	-	Current tool age CUR.TIME
		12	-	PLC status
		13	-	Tooth length in the tool axis LCUTS
		14	-	Maximum plunge angle ANGLE
		15	-	TT: Number of tool teeth CUT
		16	-	TT: Wear tolerance for length LTOL
		17	-	TT: Wear tolerance for radius RTOL
		18	-	TT: Direction of rotation DIRECT 0 = positive, –1 = negative
		19	-	TT: Offset in plane R-OFFS R = 99999.9999
		20	-	TT: Offset in length L-OFFS
		21	-	TT: Break tolerance for length LBREAK
		22	-	TT: Break tolerance for radius RBREAK
		28	-	Maximum spindle speed [rpm] NMAX
		32	-	Point angle TANGLE
		34	-	LIFTOFF allowed (0 = No, 1 = Yes)
		35	-	Wear tolerance for radius R2TOL
		36	-	Tool type TYPE (miller = 0, grinder = 1, ... touch probe = 21)
		37	-	Corresponding line in the touch-probe table
		38	-	Timestamp of last use
		39	-	ACC
		40	-	Pitch for thread cycles
		41	-	AFC: reference load
		42	-	AFC: overload early warning
		43	-	AFC: overload NC stop
		44	-	Exceeding the tool life
Read and write data of current turning tool				
	951	1	-	Tool number
		2	-	Tool length XL
		3	-	Tool length YL
		4	-	Tool length ZL
		5	-	Tool length oversize DXL
		6	-	Oversize in tool length DYL
		7	-	Tool length oversize DZL
		8	-	Tooth radius (RS)

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		9	-	Tool orientation (TO)
		10	-	Angle of spindle orientation (ORI)
		11	-	Tool angle P_ANGLE
		12	-	Point angle T_ANGLE
		13	-	Recessing width CUT_WIDTH
		14	-	Type (e.g. roughing, finishing, threading, recessing or button tool)
		15	-	Length of cutting edge CUT_LENGTH
		16	-	Compensation of workpiece diameter WPL-DX-DIAM in the working plane coordinate system WPL-CS
		17	-	Compensation of workpiece diameter WPL-DZL in the working plane coordinate system WPL-CS
		18	-	Recessing width oversize
		19	-	Cutting radius oversize

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Freely available memory area for tool management				
	956	0-9	-	Freely available data area for tool management. The data is not reset when the program is aborted.
Tool usage and tooling				
	975	1	-	Tool usage test for the current NC program: Result –2: Test not possible, function disabled in the configuration Result –1: Test not possible, tool usage file missing Result 0: Test OK, all tools available Result 1: Test not OK
		2	Line	Check availability of the tools required in the pallet from line IDX in the current pallet table. –3 = No pallet is defined in row IDX, or function was called outside of pallet editing –2 / –1 / 0 / 1 see NR1
Lift off the tool at NC stop				
	980	3	-	(This function is obsolete—HEIDENHAIN recommends not to use it any longer. ID980 NR3 = 1 is equivalent to ID980 NR1 = –1, ID980 NR3 = 0 has the same effect as ID980 NR1 = 0. Other values are not permissible.) Enable lift-off to the value defined in CfgLiftOff: 0 = Lock lift-off function 1 = Enable lift-off function
Touch probe cycles and coordinate transformations				
	990	1	-	Approach behavior: 0 = Standard behavior 1 = Approach probing position without compensation Effective radius, set-up clearance is zero
		2	16	Automatic / Manual machine operating modes
		4	-	0 = Stylus not deflected 1 = Stylus deflected
		6	-	TT tool touch probe active? 1 = Yes 0 = No
		8	-	Momentary spindle angle in [°]
		10	QS parameter no.	Determine the tool number from the tool name. The return value depends on the rules configured for the search of the replacement tool. If there are multiple tools with the same name, the first tool from the tool table will be selected.

Group name	Group number ID...	System data number NO....	Index IDX...	Description
				If the tool selected by these rules is locked, a replacement tool will be returned. –1: No tool with the specified name found in the tool table or all qualifying tools are locked.
		16	0	0 = Transfer control over the channel spindle to the PLC, 1 = Assume control over the channel spindle
			1	0 = Pass tool spindle control to the PLC, 1 = Take control of the tool spindle
		19	-	Suppress touch prove movement in cycles: 0 = Movement will be suppressed (CfgMachineSimul/simMode parameter not equal to FullOperation or **Test Run** operating mode is active) 1 = Movement will be performed (CfgMachineSimul/simMode parameter = FullOperation, can be programmed for testing purposes)
Status of execution				
	992	10	-	Block scan active 1 = yes, 0 = no
		11	-	Block scan—information on block scan: 0 = NC program started without block scan 1 = Iniprog system cycle is run before block scan 2 = Block scan is running 3 = Functions are being updated –1 = Iniprog cycle was canceled before block scan –2 = Cancellation during block scan –3 = Cancellation of the block scan after the search phase, before or during the update of functions –99 = Implicit cancellation
		12	-	Type of canceling for interrogation within the OEM_CANCEL macro: 0 = No cancellation 1 = Cancellation due to error or emergency stop 2 = Explicit cancellation with internal stop after stop in the middle of the block 3 = Explicit cancellation with internal stop after stop at the end of a block
		14	-	Number of the last FN14 error
		16	-	Real execution active? 1 = execution, 0 = simulation
		17	-	2-D graphics during programming active? 1 = yes 0 = no

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		18	-	Live programming graphics (**AUTO DRAW** soft key) active? 1 = Yes 0 = No
		20	-	Information on combined milling/turning mode of operation: 0 = Milling (after **FUNCTION MODE MILL**) 1 = Turning (after **FUNCTION MODE TURN**) 10 = Execute the operations for the turning-to-milling transition 11 = Execute the operations for the milling-to-turning transition
		30	-	Interpolation of multiple axes permitted? 0 = No (e.g. for straight cut control) 1 = yes
		31	-	R+/R– possible/permitted in MDI mode? 0 = No 1 = Yes
		32	0	Cycle call possible/permitted? 0 = No 1 = Yes
			Cycle number	Single cycle enabled: 0 = No 1 = Yes
		40	-	Copy tables in **Test Run** operating mode? Value 1 will be set when a program is selected and when the **RESET+START** soft key is pressed. The **iniprog.h** system cycle will then copy the tables and reset the system datum. 0 = no 1 = yes
		101	-	M101 active (visible condition)? 0 = no 1 = yes
		136	-	M136 active? 0 = no 1 = yes

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Activate machine parameter subfile				
	1020	13	QS parameter no.	Has a machine parameter subfile with path from QS number (IDX) been loaded? 1 = Yes 0 = No
Configuration settings for cycles				
	1030	1	-	Display **spindle does not rotate** error message? **(CfgGeoCycle/displaySpindleErr)** 0 = no, 1 = yes
			-	Check **the algebraic sign for depth error message!** display? **(CfgGeoCycle/displayDepthErr)** 0 = no, 1 = yes
Write or read PLC data synchronously in real time				
	2000	10	Marker no.	PLC markers General note for NR10 to NR80: The functions are executed synchronously in real time, i.e. the function is not executed until the corresponding point is reached in the program. HEIDENHAIN recommends using the **WRITE TO PLC** or **READ FROM PLC** commands instead of ID2000 and synchronizing the execution in real time by using **FN20: WAIT FOR SYNC**.
		20	Input no.	PLC input
		30	Output no.	PLC output
		40	Counter no.	PLC counter
		50	Timer no.	PLC timer
		60	Byte no.	PLC byte
		70	Word no.	PLC word
		80	Double-word no.	PLC double word

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Do not write or read PLC data synchronously in real time				
	2001	10-80	see ID 2000	Same as ID2000 NR10 to NR80, but not synchronous in real time. Function is executed in the look-ahead calculation. HEIDENHAIN recommends using the **WRITE TO PLC** and **READ FROM PLC** commands instead of ID2001.
Bit test				
	2300	Number	Bit number	This function checks whether a bit has been set in a number. The number to be checked is transferred as NR, the bit to be searched for as IDX, with IDX0 designating the least significant bit. To call this function for great numbers, make sure to transfer NR as a Q parameter. 0 = Bit not set 1 = Bit set
Read program information (system string)				
	10010	1	-	Path of the current main program or pallet program.
		2	-	Path of the NC program shown in the block display.
		3	-	Path of the cycle selected with **SEL CYCLE** or **CYCLE DEF 12 PGM CALL**, or path of the currently active cycle
		10	-	Path of the NC program selected with **SEL PGM "..."**.
Indexed access to QS parameters				
	10015	20	QS parameter no.	Reads QS(IDX)
		30	QS parameter no.	Returns the string that you obtain if you replace anything except for letters and digits in QS(IDX) by '_'.
Read channel data (system string)				
	10025	1	-	Name of machining channel (key)
Read data for SQL tables (system string)				
	10040	1	-	Symbolic name of the preset table.
		2	-	Symbolic name of the datum table.
		3	-	Symbolic name of the pallet preset table.
		10	-	Symbolic name of the tool table.
		11	-	Symbolic name of the pocket table.
		12	-	Symbolic name of the turning tool table

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Values programmed in the tool call (system string)				
	10060	1	-	Tool name
Read machine kinematics (system strings)				
	10290	10	-	Symbolic name of the machine kinematics from Channels/ChannelSettings/CfgKinList/kinCompositeModels programmed in **FUNCTION MODE MILL** or **FUNCTION MODE TURN**.
Traverse range switchover (system string)				
	10300	1	-	Key name of the last active range of traverse
Read current system time (system string)				
	10321	1 - 16	-	1: DD.MM.YYYY hh:mm:ss 2 and 16: DD.MM.YYYY hh:mm 3: DD.MM.YY hh:mm 4: YYYY-MM-DD hh:mm:ss 5 and 6: YYYY-MM-DD hh:mm 7: YY-MM-DD hh:mm 8 and 9: DD.MM.YYYY 10: DD.MM.YY 11: YYYY-MM-DD 12: YY-MM-DD 13 and 14: hh:mm:ss 15: hh:mm As an alternative, you can use **DAT** in **SYSSTR(...)** to specify a system time in seconds that is to be used for formatting.
Read data of touch probes (TS, TT) (system string)				
	10350	50	-	TS probe type from TYPE column of the touch probe table (**tchprobe.tp**)
		70	-	Type of TT tool touch probe from CfgTT/type.
		73	-	Key name of the active tool touch probe TT from **CfgProbes/activeTT**.
Read and write data of touch probes (TS, TT) (system string)				
	10350	74	-	Serial number of the active tool touch probe TT from **CfgProbes/activeTT**.
Read the data for pallet machining (system string)				
	10510	1	-	Pallet name
		2	-	Path of the selected pallet table.
Read version ID of the NC software (system string)				
	10630	10	-	The string corresponds to the format of the version ID shown, e.g. **340590 09** or **817601 05 SP1**.
Read information on unbalance cycle (system string)				
	10855	1	-	Path of the unbalance calibration table belonging to the active kinematics

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Read data of the current tool (system string)				
	10950	1	-	Current tool name
		2	-	Entry from the DOC column of the active tool
		3	-	AFC control setting
		4	-	Tool-carrier kinematics
		5	-	Entry from the DR2TABLE column – file name of the compensation value table for 3D-ToolComp

Comparison: FN 18 functions

The following table lists the FN 18 functions from previous controls, which were not implemented in this manner in the TNC 640.

In most cases, this function has been replaced by another function.

No.	IDX	Contents	Replacement function
ID 10 Program information			
1	-	mm/inch condition	Q113
2	-	Overlap factor for pocket milling	CfgRead
4	-	Number of the active fixed cycle	ID 10 no. 3
ID 20 Machine status			
15	Log. axis	Assignment between logic and geometric axes	
16	-	Feed rate for transition arcs	
17	-	Currently selected range of traverse	SYSTRING 10300
19	-	Maximum spindle speed for current gear stage and spindle	Maximum gear range: ID 90 No. 2
ID 50 Data from the tool table			
23	Tool no.	PLC value	1)
24	Tool no.	Probe center offset in reference axis (CAL-OF1)	ID 350 NR 53 IDX 1
25	Tool no.	Probe center offset in minor axis (CALOF-2)	ID 350 NR 53 IDX 2
26	Tool no.	Spindle angle during calibration (CAL-ANG)	ID 350 NR 54
27	Tool no.	Tool type for pocket table (PTYP)	2)
29	Tool no.	Position P1	1)
30	Tool no.	Position P2	1)
31	Tool no.	Position P3	1)
33	Tool no.	Thread pitch (Pitch)	ID 50 NR 40
ID 51 Data from the pocket table			
6	Pocket no.	Tool type	2)
7	Pocket no.	P1	2)

No.	IDX	Contents	Replacement function
8	Pocket no.	P2	2)
9	Pocket no.	P3	2)
10	Pocket no.	P4	2)
11	Pocket no.	P5	2)
12	Pocket no.	Pocket reserved 0 = No, 1 = Yes	2)
13	Pocket no.	Box magazine: Pocket above occupied: 0 = No, 1 = Yes	2)
14	Pocket no.	Box magazine: Pocket below occupied: 0 = No, 1 = Yes	2)
15	Pocket no.	Box magazine: Pocket to the left occupied: 0 = No, 1 = Yes	2)
16	Pocket number	Box magazine: Pocket to the right occupied: 0 = No, 1 = Yes	2)
ID 56 File information			
1	-	Number of lines of the tool table	
2	-	Number of lines of the active datum table	
3	Q parameters	Number of active axes that are programmed in the active datum table	
4	-	Number of lines in a freely definable table that has been opened with FN 26: TABOPEN	
ID 214 Current contour data			
1	-	Contour transition mode	
2	-	Max. linearization error	
3	-	Mode for M112	
4	-	Character mode	
5	-	Mode for M124	1)
6	-	Specification for contour pocket machining	
7	-	Filter for control loop	
8	-	Tolerance programmed with Cycle 32 or MP 1096	ID 30 no. 48
ID 240 Nominal positions in the REF system			
8	-	ACTUAL position in the REF system	
ID 280 Information on M128			
2	-	Feed rate that was programmed with M128	ID 280 NR 3
ID 290 Switch the kinematics			
1	-	Line of the active kinematics table	SYSSTRING 10290
2	Bit no.	Interrogate the bits in MP7500	Cfgread
3	-	Status of collision monitoring (old)	Can be activated and deactivated in the NC program

No.	IDX	Contents	Replacement function
4	-	Status of collision monitoring (new)	Can be activated and deactivated in the NC program
ID 310 Modifications of geometrical behavior			
116	-	M116: –1 = On, 0 = Off	
126	-	M126: –1 = On, 0 = Off	
ID 350 Touch-probe data			
10	-	TS: Touch-probe axis	ID 20 NR 3
11	-	TS: Effective ball radius	ID 350 NR 52
12	-	TS: Effective length	ID 350 NR 51
13	-	TS: Ring gauge radius	
14	1/2	TS: Center offset in reference/minor axis	ID 350 NR 53
15	-	TS: Direction of center offset relative to 0° position	ID 350 NR 54
20	1/2/3	TT: Center point X/Y/Z	ID 350 NR 71
21	-	TT: Plate radius	ID 350 NR 72
22	1/2/3	TT: 1st probing position X/Y/Z	Cfgread
23	1/2/3	TT: 2nd probing position X/Y/Z	Cfgread
24	1/2/3	TT: 3rd probing position X/Y/Z	Cfgread
25	1/2/3	TT: 4th probing position X/Y/Z	Cfgread
ID 370 Touch probe cycle settings			
1	-	Do not move to set-up clearance in Cycle 0.0 and 1.0 (as with ID990 NR1)	ID 990 NR 1
2	-	MP 6150 Rapid traverse for measurement	ID 350 NR 55 IDX 1
3	-	MP 6151 Machine rapid traverse as rapid traverse for measurement	ID 350 NR 55 IDX 3
4	-	MP 6120 Feed rate for measurement	ID 350 NR 55 IDX 2
5	-	MP 6165 Angle tracking on/off	ID 350 NR 57
ID 501 Datum table (REF system)			
Line	Column	Value in datum table	Preset table
ID 502 Preset table			
Line	Column	Read the value from preset table, taking into account the active machining system	
ID 503 Preset table			
Line	Column	Read the value directly from the preset table	ID 507
ID 504 Preset table			
Line	Column	Read the basic rotation from the preset table	ID 507 IDX 4-6
ID 505 Datum table			
1	-	0 = No datum table selected 1 = Datum table selected	

No.	IDX	Contents	Replacement function
ID 510 Data for pallet machining			
7	-	Test the insertion of a fixture from the PAL line	
ID 530 Active preset			
2	Line	Write-protect the line in the active preset table: 0 = No, 1 = Yes	FN 26 and FN 28: Read Locked column
ID 990 Approach behavior			
2	10	0 = No execution in block scan 1 = Execution in block scan	ID 992 NR 10 / NR 11
3	Q parameters	Number of axes that are programmed in the selected datum table	
ID 1000 Machine parameter			
MP number	MP index	Value of the machine parameter	CfgRead
ID 1010 Machine parameter is defined			
MP number	MP index	0 = Machine parameter does not exist 1 = Machine parameter exists	CfgRead

1) Function or table column no longer exists

2) Use FN 26 and FN 28 or SQL to read out the table cell

17.2 Overview tables

Miscellaneous functions

M	Effect	Effective at block Start	End	Page
M0	Program STOP/Spindle STOP/Coolant OFF		■	225
M1	Optional program run STOP/Spindle STOP/Coolant OFF		■	225
M2	Stop program/Spindle STOP/Coolant OFF/ CLEAR status display (depending on machine parameter)/Return jump to block 1		■	225
M3 M4 M5	Spindle ON clockwise Spindle ON counterclockwise Spindle STOP	■ ■ 	 ■	225
M6	Tool change/STOP program run (depending on machine parameter)/Spindle STOP		■	225
M8 M9	Coolant ON Coolant OFF	■ 	 ■	225
M13 M14	Spindle ON clockwise/Coolant ON Spindle ON counterclockwise/Coolant on	■ ■		225
M30	Same function as M2		■	225
M89	Vacant miscellaneous function **or** cycle call, modally effective (depending on machine parameter)	■ 	 ■	Cycles Manual
M91	Within the positioning block: Coordinates are referenced to machine datum	■		226
M92	Within the positioning block: Coordinates are referenced to a position defined by machine manufacturer, e.g. tool change position	■		226
M94	Reduce the rotary axis display to a value below 360°	■		437
M97	Machine small contour steps		■	229
M98	Machine open contours completely		■	230
M99	Blockwise cycle call		■	Cycles Manual
M101 M102	Automatic tool change with replacement tool if maximum tool life has expired Reset M101		■ ■	130
M103	Feed rate factor for plunging movements	■		231
M107 M108	Suppress error message for replacement tools with oversize Reset M107		■ ■	450
M109 **M110** M111	Constant contouring speed at cutting edge (feed rate increase andreduction) Constant contouring speed at cutting edge (only feed rate reduction) Reset M109/M110	■ ■ 	 ■	232
M116 M117	Feed rate in mm/min on rotary axes Reset M116	■ 	 ■	435
M118	Superimpose handwheel positioning during program run	■		235
M120	Pre-calculate the radius-compensated contour (LOOK AHEAD)	■		233
M126 M127	Shorter-path traverse of rotary axes Reset M126	■ 	 ■	436

M	Effect	Effective at block: Start	End	Page
M128	Maintaining the position of the tool tip when positioning with tilted axes (TCPM)	■		438
M129	Reset M128		■	
M130	Within the positioning block: Points are referenced to the untilted coordinate system	■		228
M136	Feed rate F in millimeters per spindle revolution	■		232
M137	Reset M136			
M138	Selection of tilted axes	■		441
M140	Retraction from the contour in the tool-axis direction	■		237
M141	Suppress touch probe monitoring	■		239
M143	Delete basic rotation	■		240
M144	Compensating the machine's kinematic configuration for ACTUAL/NOMINAL positions at end of block	■		442
M145	Reset M144		■	
M148	Automatically retract tool from the contour at an NC stop	■		240
M149	Reset M148		■	
M197	Corner rounding	■	■	241

User functions

User functions		
Short description	■	Basic version: 3 axes plus closed-loop spindle
	■	Fourth NC axis plus auxiliary axis
		or
	□	8 additional axes or 7 additional axes plus 2nd spindle
	■	Digital current and speed control
Program entry	In HEIDENHAIN conversational format and DIN/ISO	
Position entry	■	Nominal positions for lines and arcs in Cartesian coordinates or polar coordinates
	■	Incremental or absolute dimensions
	■	Display and entry in mm or inches
Tool compensation	■	Tool radius in the working plane and tool length
	■	Radius-compensated contour look-ahead for up to 99 NC blocks (M120)
	2	Three-dimensional tool-radius compensation for changing tool data without having to recalculate an existing NC program
Tool tables	Multiple tool tables with any number of tools	
Constant contour speed	■	With respect to the path of the tool center
	■	With respect to the cutting edge
Parallel operation	Creating an NC program with graphical support while another NC program is being run	
3-D machining (Advanced Function Set 2)	**2**	Motion control with minimum jerk
	2	3-D tool compensation through surface-normal vectors
	2	Using the electronic handwheel to change the angle of the swivel head during program run without affecting the position of the tool center point (tool tip or center of sphere) (TCPM = **T**ool **C**enter **P**oint **M**anage-ment)
	2	Keeping the tool normal to the contour
	2	Tool radius compensation perpendicular to traversing and tool direction
Rotary table machining (Advanced Function Set 1)	**1**	Programming of cylindrical contours as if in two axes
	1	Feed rate in distance per minute

User functions		
Contour elements	■	Straight line
	■	Chamfer
	■	Circular path
	■	Circle center
	■	Circle radius
	■	Tangentially connected arc
	■	Rounded corners
Approaching and departing the contour	■	Via straight line: tangential or perpendicular
	■	Via circular arc
FK free contour programming	■	FK free contour programming in HEIDENHAIN conversational format with graphic support for workpiece drawings not dimensioned for NC
Program jumps	■	Subprograms
	■	Program section repeats
	■	External NC programs
Machining cycles	■	Cycles for drilling, and conventional and rigid tapping
	■	Roughing of rectangular and circular pockets
	■	Cycles for pecking, reaming, boring, and counterboring
	■	Cycles for milling internal and external threads
	■	Finishing of rectangular and circular pockets
	■	Cycles for clearing level and inclined surfaces
	■	Cycles for milling linear and circular slots
	■	Cartesian and polar point patterns and point patterns for DataMatrix code
	■	Contour-parallel contour pocket
	■	Contour train
	■	Cycles for turning operations
	■	OEM cycles (special cycles developed by the machine manufacturer) can also be integrated
Coordinate transformation	■	Datum shift, rotation, mirroring
	■	Scaling factor (axis-specific)
	1	Tilting the working plane (Advanced Function Set 1)

User functions	
Q parameters Programming with variables	■ Mathematical functions: =, +, –, *, sin α, cos α, root ■ Logical operations (=, ≠, <, >) ■ Calculating with parentheses ■ tan α, arc sin, arc cos, arc tan, a^n, e^n, ln, log, absolute value of a number, constant π, negation, truncation of digits before or after the decimal point ■ Functions for calculation of circles ■ String parameters
Programming aids	■ Calculator ■ Color highlighting of syntax elements ■ Complete list of all current error messages ■ Context-sensitive help function for error messages ■ Graphic support for the programming of cycles ■ Comment blocks in NC program
Teach-In	■ Actual positions can be transferred directly to the NC program
Test graphics Display modes	■ Graphic simulation before a program run, even while another NC program is being run ■ Plan view / projection in 3 planes / 3-D view / 3-D line graphic ■ Detail enlargement
Programming graphics	■ In the Programming mode, the contours of the NC blocks are drawn on screen while they are being entered (2-D pencil-trace graphics), even while another NC program is being run
Program-run graphics Display modes	■ Graphic simulation of real-time machining in plan view / projection in 3 planes / 3-D view
Machining time	■ Calculation of machining time in the **Test Run** operating mode ■ Display of the current machining time in the Program Run operating modes
Contour, returning to	■ Block scan in any NC block in the NC program, returning the tool to the calculated nominal position to continue machining ■ NC program interruption, contour departure and return
Datum tables	■ Multiple datum tables for storing workpiece-specific datums
Touch probe cycles	■ Calibrating the touch probe ■ Compensation of workpiece misalignment, manual or automatic ■ Presetting, manual or automatic ■ Automatically measuring workpieces ■ Cycles for automatic tool measurement ■ Cycles for automatic kinematics measurement

17.3 Differences between the TNC 640 and the iTNC 530

Comparison: PC software

Function	TNC 640	iTNC 530
M3D Converter for the creation of high-resolution collision objects for collision monitoring (DCM)	Available	Not available
ConfigDesign for the configuration of machine parameters	Available	Not available
TNCanalyzer for the analysis and evaluation of service files	Available	Not available

Comparison: User functions

Function	TNC 640	iTNC 530
Program entry		
■ smarT.NC	■ –	■ X
■ ASCII editor	■ X, directly editable	■ X, editable after conversion
Position entry		
■ Set the last tool position as pole (empty CC block)	■ X (error message if pole transfer is ambiguous)	■ X
■ Spline sets (**SPL**)	■ –	■ X, with option 9
Tool table		
■ Flexible management of tool types	■ X	■ –
■ Filtered display of selectable tools	■ X	■ –
■ Sorting function	■ X	■ –
■ Column names	■ Sometimes with _	■ Sometimes with **-**
■ Form view	■ Switchover with Screen Layout key	■ Switchover by soft key
■ Exchange of tool table between TNC 640 and iTNC 530	■ X	■ Not possible
Touch probe table for managing different 3-D touch probes	X	–
Cutting data calculator: Automatic calculation of spindle speed and feed rate	■ Simple cutting data calculator without stored table ■ Cutting data calculator with stored technology tables	Using stored technology tables

Function	TNC 640	iTNC 530
Define any tables	■ Freely definable tables (.TAB files) ■ Reading and writing with FN functions ■ Definable via config. data ■ The names of tables and table columns must start with a letter, and no arithmetic operators are permitted ■ Reading and writing with SQL functions	■ Freely definable tables (.TAB files) ■ Reading and writing with FN functions
Traverse in tool-axis direction		
■ Manual operation (3-D ROT menu)	■ X	■ X, FCL2 function
■ With handwheel superimpositioning	■ X	■ X, option 44
Entry of feed rates:		
■ **FT** (time in seconds for path)	■ –	■ X
■ **FMAXT** (only for active rapid traverse potentiometer: time in seconds for path)	■ –	■ X
FK free contour programming		
■ Conversion of FK program to Klartext conversational language	■ –	■ X
■ FK blocks in combination with **M89**	■ –	■ X
Program jumps:		
■ Maximum number of labels	■ 65535	■ 1000
■ Subprograms	■ X	■ X
■ Nesting depth for subprograms	■ 20	■ 6
Q parameter programming:		
■ FN 15: PRINT	■ –	■ X
■ FN 25: PRESET	■ –	■ X
■ FN 29: PLC LIST	■ X	■ –
■ FN31: RANGE SELECT	■ –	■ X
■ FN32: PLC PRESET	■ –	■ X
■ FN37: EXPORT	■ X	■ –
■ Write to LOG file with **FN16**	■ X	■ –
■ Displaying parameter contents in the additional status display	■ X	■ –
■ **SQL** functions for writing and reading tables	■ X	■ –

Function	TNC 640	iTNC 530
Graphic support		
■ 2-D programming graphics	■ X	■ X
■ REDRAW function (**REDRAW**)	■ –	■ X
■ Show grid lines as the background	■ X	■ –
■ Test graphics (plan view, projection on 3 planes, 3-D view)	■ X	■ X
■ Coordinates of line intersection for projection in 3 planes	■ –	■ X
■ Factor in tool change macro	■ X (differing to actual execution)	■ X
Preset table		
■ Line 0 of the preset table can be edited manually	■ X	■ –
Programming aids:		
■ Color highlighting of syntax elements	■ X	■ –
■ Calculator	■ X (scientific)	■ X (standard)
■ Convert NC blocks to comments	■ X	■ –
■ Structure blocks in NC program	■ X	■ X
■ Structure view in test run	■ –	■ X
Dynamic Collision Monitoring (DCM):		
■ Fixture monitoring	■ –	■ X, option 40
■ Tool carrier management	■ X	■ X, option 40

Function	TNC 640	iTNC 530
CAM support:		
■ Load contours from Step data and Iges data	■ X, option 42	■ –
■ Load machining positions from Step data and Iges data	■ X, option 42	■ –
■ Offline filter for CAM files	■ –	■ X
■ Stretch filter	■ X	■ –
MOD functions:		
■ User parameters	■ Config data	■ Numerical structure
■ OEM help files with service functions	■ –	■ X
■ Data medium inspection	■ –	■ X
■ Load service packs	■ –	■ X
■ Specify the axes for actual position capture	■ –	■ X
■ Configure counter	■ X	■ –
Special functions:		
■ Create reverse program	■ –	■ X
■ Define the counter with **FUNCTION COUNT**	■ X	■ –
■ Define the dwell time with **FUNCTION FEED**	■ X	■ –
■ Define the dwell time with **FUNCTION DWELL**	■ X	■ –
■ Determine the integration of the programmed coordinates with **FUNCTION PROG PATH**	■ X	■ –
Status displays:		
■ Dynamic display of Q-parameter contents, definable number ranges	■ X	■ –
■ Graphic display of residual run time	■ –	■ X
Individual color settings of user interface	–	X

Comparison: Miscellaneous functions

M	Effect	TNC 640	iTNC 530
M00	Program STOP/Spindle STOP/Coolant OFF	X	X
M01	Optional program STOP	X	X
M02	Stop program/Spindle STOP/Coolant OFF/ Clear status display (depending on machine parameter)/Return jump to block 1	X	X
M03 M04 M05	Spindle ON clockwise Spindle ON counterclockwise Spindle STOP	X	X
M06	Tool change/Program run STOP (machine-specific function)/ Spindle STOP	X	X
M08 M09	Coolant ON Coolant OFF	X	X
M13 M14	Spindle ON clockwise/Coolant ON Spindle ON counterclockwise/Coolant on	X	X
M30	Same function as M02	X	X
M89	Free miscellaneous function **or** cycle call, modally effective (machine-specific function)	X	X
M90	Constant contouring speed at corners (not required at TNC 640)	–	X
M91	Within the positioning block: Coordinates are referenced to machine datum	X	X
M92	Within the positioning block: Coordinates are referenced to a position defined by machine manufacturer, e.g. tool change position	X	X
M94	Reduce the rotary axis display to a value below 360°	X	X
M97	Machine small contour steps	X	X
M98	Machine open contours completely	X	X
M99	Blockwise cycle call	X	X
M101 M102	Automatic tool change with replacement tool if maximum tool life has expired Reset M101	X	X
M103	Reduce feed rate during plunging to factor F (percentage)	X	X
M104	Reactivate most recently set preset	– (recommended: Cycle 247)	X
M105 M106	Machining with second k_v factor Machining with first k_v factor	–	X
M107 M108	Suppress error message for replacement tools with oversize Reset M107	X	X
M109 **M110** M111	Constant contouring speed at cutting edge (feed rate increase and reduction) Constant contouring speed at cutting edge (only feed rate reduction) Reset M109/M110	X	X
M112 M113	Enter contour transitions between any two contour transitions Reset M112	– (recommended: Cycle 32)	X

M	Effect	TNC 640	iTNC 530
M114 M115	Automatic compensation of machine geometry when working with tilted axes Reset M114	– (recommended: M128, TCPM)	X, option 8
M116 M117	Feed rate on rotary tables in mm/min Reset M116	X, option 8	X, option 8
M118	Superimpose handwheel positioning during program run	X	X
M120	Pre-calculate the radius-compensated contour (LOOK AHEAD)	X	X
M124	Contour filter	– (possible via user parameters)	X
M126 M127	Shorter-path traverse of rotary axes Reset M126	X	X
M128 M129	Maintaining the position of the tool tip when positioning tilted axes (TCPM) Reset M128	X, option 9	X, option 9
M130	Within the positioning block: Points are referenced to the untilted coordinate system	X	X
M134 M135	Precision stop at non-tangential contour transitions when positioning with rotary axes Reset M134	X (depends on the machine tool builder)	X
M136 M137	Feed rate F in millimeters per spindle revolution Reset M136	X	X
M138	Selection of tilted axes	X	X
M140	Retraction from the contour in the tool-axis direction	X	X
M141	Suppress touch probe monitoring	X	X
M142	Delete modal program information	–	X
M143	Delete basic rotation	X	X
M144 M145	Compensating the machine's kinematic configuration for ACTUAL/NOMINAL positions at end of block Reset M144	X, option 9	X, option 9
M148 M149	Automatically retract tool from the contour at an NC stop Reset M148	X	X
M150	Suppress limit switch message	– (possible via FN 17)	X
M197	Rounding the corners	X	–
M200 - M204	Laser cutting functions	–	X

Comparator: Cycles

Cycle	TNC 640	iTNC 530
1 **PECKING** (recommended: Cycle 200, 203, 205)	–	X
2 **TAPPING** (recommended: Cycle 206, 207, 208)	–	X
3 **SLOT MILLING** (recommended: Cycle 253)	–	X
4 **POCKET MILLING** (recommended: Cycle 251)	–	X
5 **CIRCULAR POCKET** (recommended: Cycle 252)	–	X
6 **ROUGH-OUT** (SL I, recommended: SL II, Cycle 22)	–	X
7 **DATUM SHIFT**	X	X
8 **MIRROR IMAGE**	X	X
9 **DWELL TIME**	X	X
10 **ROTATION**	X	X
11 **SCALING**	X	X
12 **PGM CALL**	X	X
13 **ORIENTATION**	X	X
14 **CONTOUR GEOMETRY**	X	X
15 **PILOT DRILLING** (SL I, recommended: SL II, Cycle 21)	–	X
16 **CONTOUR MILLING** (SL I, recommended: SL II, Cycle 24)	–	X
17 **RIGID TAPPING** (recommended: Cycle 207, 209)	–	X
18 **THREAD CUTTING**	X	X
19 **WORKING PLANE**	X, option 8	X, option 8
20 **CONTOUR DATA**	X	X
21 **PILOT DRILLING**	X	X
22 **ROUGH-OUT**	X	X
23 **FLOOR FINISHING**	X	X
24 **SIDE FINISHING**	X	X
25 **CONTOUR TRAIN**	X	X
26 **AXIS-SPECIFIC SCALING**	X	X
27 **CYLINDER SURFACE**	X, option 8	X, option 8
28 **CYLINDER SURFACE**	X, option 8	X, option 8
29 **CYL SURFACE RIDGE**	X, option 8	X, option 8
30 **RUN CAM DATA**	–	X
32 **TOLERANCE**	X	X
39 **CYL. SURFACE CONTOUR**	X, option 8	X, option 8
200 **DRILLING**	X	X
201 **REAMING**	X	X
202 **BORING**	X	X
203 **UNIVERSAL DRILLING**	X	X
204 **BACK BORING**	X	X

Cycle	TNC 640	iTNC 530
205 **UNIVERSAL PECKING**	X	X
206 **TAPPING**	X	X
207 **RIGID TAPPING**	X	X
208 **BORE MILLING**	X	X
209 **TAPPING W/ CHIP BRKG**	X	X
210 **SLOT RECIP. PLNG** (recommended: Cycle 253)	–	X
211 **CIRCULAR SLOT** (recommended: Cycle 254)	–	X
212 **POCKET FINISHING** (recommended: Cycle 251)	–	X
213 **STUD FINISHING** (recommended: Cycle 256)	–	X
214 **C. POCKET FINISHING** (recommended: Cycle 252)	–	X
215 **C. STUD FINISHING** (recommended: Cycle 257)	–	X
220 **POLAR PATTERN**	X	X
221 **CARTESIAN PATTERN**	X	X
224 **DATAMATRIX CODE PATTERN**	X	–
225 **ENGRAVING**	X	X
230 **MULTIPASS MILLING** (recommended: Cycle 233)	–	X
231 **RULED SURFACE**	–	X
232 **FACE MILLING**	X	X
233 **FACE MILLING**	X	–
238 **MEASURE MACHINE STATUS**	X, Option 155	–
239 **ASCERTAIN THE LOAD**	X, option 143	–
240 **CENTERING**	X	X
241 **SINGLE-LIP D.H.DRLNG**	X	X
247 **PRESETTING**	X	X
251 **RECTANGULAR POCKET**	X	X
252 **CIRCULAR POCKET**	X	X
253 **SLOT MILLING**	X	X
254 **CIRCULAR SLOT**	X	X
256 **RECTANGULAR STUD**	X	X
257 **CIRCULAR STUD**	X	X
258 **POLYGON STUD**	X	–
262 **THREAD MILLING**	X	X
263 **THREAD MLLNG/CNTSNKG**	X	X
264 **THREAD DRILLNG/MLLNG**	X	X
265 **HEL. THREAD DRLG/MLG**	X	X
267 **OUTSIDE THREAD MLLNG**	X	X
270 **CONTOUR TRAIN DATA** for defining the behavior of Cycle 25	X	X
271**OCM CONTOUR DATA**		–

Cycle	TNC 640	iTNC 530
272 **OCM ROUGHING**		–
273 **OCM FINISHING FLOOR**		–
274 **OCM FINISHING SIDE**		–
275 **TROCHOIDAL SLOT**	X	X
276 **THREE-D CONT. TRAIN**	X	X
285 **DEFINE GEAR**	X, Option 157	–
286 **GEAR HOBBING**	X, Option 157	–
287 **GEAR SKIVING**	X, Option 157	–
290 **INTERPOLATION TURNING**	–	X, option 96
291 **COUPLG.TURNG.INTERP.**	X, option 96	–
292 **CONTOUR.TURNG.INTRP.**	X, option 96	–
800 **ADJUST XZ SYSTEM**	X, Option 50	–
801 **RESET ROTARY COORDINATE SYSTEM**	X, Option 50	–
810 **TURN CONTOUR LONG.**	X, Option 50	–
811 **SHOULDER, LONGITDNL.**	X, Option 50	–
812 **SHOULDER, LONG. EXT.**	X, Option 50	–
813 **TURN PLUNGE CONTOUR LONGITUDINAL**	X, Option 50	–
814 **TURN PLUNGE LONGITUDINAL EXT.**	X, Option 50	–
815 **CONTOUR-PAR. TURNING**	X, Option 50	–
820 **TURN CONTOUR TRANSV.**	X, Option 50	–
821 **SHOULDER, FACE**	X, Option 50	–
822 **SHOULDER, FACE. EXT.**	X, Option 50	–
823 **TURN TRANSVERSE PLUNGE**	X, Option 50	–
824 **TURN PLUNGE TRANSVERSE EXT.**	X, Option 50	–
830 **THREAD CONTOUR-PARALLEL**	X, Option 50	–
831 **THREAD LONGITUDINAL**	X, Option 50	–
832 **THREAD EXTENDED**	X, Option 50	–
840 **RECESS TURNG, RADIAL**	X, Option 50	–
841 **SIMPLE REC. TURNG., RADIAL DIR.**	X, Option 50	–
842 **ENH.REC.TURNNG, RAD.**	X, Option 50	–
850 **RECESS TURNG, AXIAL**	X, Option 50	–
851 **SIMPLE REC TURNG, AX**	X, Option 50	–
852 **ENH.REC.TURNING, AX.**	X, Option 50	–
860 **CONT. RECESS, RADIAL**	X, Option 50	–
861 **SIMPLE RECESS, RADL.**	X, Option 50	–
862 **EXPND. RECESS, RADL.**	X, Option 50	–
870 **CONT. RECESS, AXIAL**	X, Option 50	–
871 **SIMPLE RECESS, AXIAL**	X, Option 50	–

Cycle	TNC 640	iTNC 530
872 **EXPND. RECESS, AXIAL**	X, Option 50	–
880 **GEAR HOBBING**	X, Option 50, Option 131	–
883 **TURNING SIMULTANEOUS FINISHING**	X, Option 50, Option 158	–
892 **CHECK UNBALANCE**	X,, Option 50	–
1000 **DEFINE RECIP. STROKE**	X, Option 156	–
1001 **START RECIP. STROKE**	X, Option 156	–
1002 **STOP RECIP. STROKE**	X, Option 156	–
1010 **DRESSING DIAMETER**	X, Option 156	–
1015 **PROFILABRICHTEN**	X, Option 156	–
1030 **ACTIVATE WHEEL EDGE**	X, Option 156	–
1032 **GRINDING WHL LENGTH COMPENSATION**	X, Option 156	–
1033 **GRINDING WHL RADIUS COMPENSATION**	X, Option 156	–

Comparison: Touch probe cycles in the Manual operation and Electronic handwheel operating modes

Cycle	TNC 640	iTNC 530
Touch-probe table for managing 3-D touch probes	X	–
Calibrating the effective length	X	X
Calibrating the effective radius	X	X
Measuring a basic rotation using a line	X	X
Setting the preset on any axis	X	X
Setting a corner as preset	X	X
Setting a circle center as preset	X	X
Setting a center line as preset	X	X
Measuring a basic rotation using two holes/cylindrical studs	X	X
Setting the preset using four holes/cylindrical studs	X	X
Setting the circle center using three holes/cylindrical studs	X	X
Determine and offset misalignment of a plane	X	–
Support of mechanical touch probes by manually capturing the current position	By soft key or hard key	By hard key
Write measurement values to the preset table	X	X
Write measurement values to the datum table	X	X

Comparison: Probing system cycles for automatic workpiece control

Cycle	TNC 640	iTNC 530
0 **REF. PLANE**	X	X
1 **POLAR PRESET**	X	X
2 **CALIBRATE TS**	–	X
3 **MEASURING**	X	X
4 **MEASURING IN 3-D**	X	X
9 **CALIBRATE TS LENGTH**	–	X
30 **CALIBRATE TT**	X	X
31 **CAL. TOOL LENGTH**	X	X
32 **CAL. TOOL RADIUS**	X	X
33 **MEASURE TOOL**	X	X
400 **BASIC ROTATION**	X	X
401 **ROT OF 2 HOLES**	X	X
402 **ROT OF 2 STUDS**	X	X
403 **ROT IN ROTARY AXIS**	X	X
404 **SET BASIC ROTATION**	X	X
405 **ROT IN C AXIS**	X	X
408 **SLOT CENTER PRESET**	X	X
409 **RIDGE CENTER PRESET**	X	X
410 **PRESET INSIDE RECTAN**	X	X
411 **PRESET OUTS. RECTAN**	X	X
412 **PRESET INSIDE CIRCLE**	X	X
413 **PRESET OUTS. CIRCLE**	X	X
414 **PRESET OUTS. CORNER**	X	X
415 **PRESET INSIDE CORNER**	X	X
416 **PRESET CIRCLE CENTER**	X	X
417 **PRESET IN TS AXIS**	X	X
418 **PRESET FROM 4 HOLES**	X	X
419 **PRESET IN ONE AXIS**	X	X
420 **MEASURE ANGLE**	X	X
421 **MEASURE HOLE**	X	X
422 **MEAS. CIRCLE OUTSIDE**	X	X
423 **MEAS. RECTAN. INSIDE**	X	X
424 **MEAS. RECTAN. OUTS.**	X	X
425 **MEASURE INSIDE WIDTH**	X	X
426 **MEASURE RIDGE WIDTH**	X	X
427 **MEASURE COORDINATE**	X	X

Cycle	TNC 640	iTNC 530
430 **MEAS. BOLT HOLE CIRC**	X	X
431 **MEASURE PLANE**	X	X
440 **MEASURE AXIS SHIFT**	–	X
441 **FAST PROBING**	X	X
444 **PROBING IN 3-D**	X, option 92	–
450 **SAVE KINEMATICS**	X, Option 48	X, option 48
451 **MEASURE KINEMATICS**	X, Option 48	X, option 48
452 **PRESET COMPENSATION**	X, Option 48	X, option 48
453 **KINEMATICS GRID**	X, Option 48, Option 52	–
460 **CALIBRATION OF TS ON A SPHERE**	X	X
461 **TS CALIBRATION OF TOOL LENGTH**	X	X
462 **CALIBRATION OF A TS IN A RING**	X	X
463 **TS CALIBRATION ON STUD**	X	X
480 **CALIBRATE TT**	X	X
481 **CAL. TOOL LENGTH**	X	X
482 **CAL. TOOL RADIUS**	X	X
483 **MEASURE TOOL**	X	X
484 **CALIBRATE IR TT**	X	X
600 **GLOBAL WORKING SPACE**	X, option 136	–
601 **LOCAL WORKING SPACE**	X, option 136	–
1410 **PROBING ON EDGE**	X	–
1411 **PROBING TWO CIRCLES**	X	–
1420 **PROBING IN PLANE**	X	–

Comparison: Differences in programming

Function	TNC 640	iTNC 530
File management:		
■ Entry of name	■ Opens **Select file** pop-up window**Select file**	■ Synchronizes the cursor
■ Support of key combinations	■ Not available	■ Available
■ Favorites Management	■ Not available	■ Available
■ Configuration of column structure	■ Not available	■ Available
Selecting a tool from the table	Selection via split-screen menu	Selection in a pop-up window
Programming special functions with the **SPEC FCT** key	Pressing the key opens a soft-key row as a submenu. To exit the submenu, press the **SPEC FCT** key again; then the control shows the last active soft-key row	Pressing the key adds the soft-key row as the last row. To exit the menu, press the **SPEC FCT** key again; then the control shows the last active soft-key row
Programming approach and departure motions with the **APPR DEP** key	Pressing the key opens a soft-key row as a submenu. To exit the submenu, press the **APPR DEP** key again; then the control shows the last active soft-key row	Pressing the key adds the soft-key row as the last row. To exit the menu, press the **APPR DEP** key again; then the control shows the last active soft-key row
Pressing the hard key **END** with active **CYCLE DEF** and **TOUCH PROBE** menus	Terminates the editing process and calls the file manager	Exits the respective menu
Calling the file manager if **CYCLE DEF** and **TOUCH PROBE** menus are active	Terminates the editing process and calls the file manager. The respective soft-key row remains selected when the file manager is exited	**Key non-functional** error message
Calling the file manager if **CYCL CALL**, **SPEC FCT**, **PGM CALL** and **APPR DEP** menus are open	Terminates the editing process and calls the file manager. The respective soft-key row remains selected when the file manager is exited	Terminates the editing process and calls the file manager. The basic soft-key row is selected when the file manager is exited

Function	TNC 640	iTNC 530
Datum table:		
■ Sorting function by values within an axis	■ Available	■ Not available
■ Resetting the table	■ Available	■ Not available
■ Switching the list/form view	■ Switch via the screen layout key	■ Switchover by toggle soft key
■ Inserting individual line	■ Allowed everywhere, renumbering possible after request. Empty line is inserted, must be filled with zeros manually	■ Only allowed at the end of the table. Line with value 0 in all columns is inserted
■ Transfer of actual position values on individual axis to the datum table using the keys	■ Available in the **Program Run Single Block** and **Program Run, Full Sequence** operating modes	■ Available
■ Transfer of actual position values on all active axes to the datum table using the keys	■ Not available	■ Available
■ Capturing the last positions measured by TS using the keys	■ Not available	■ Available
FK free contour programming:		
■ Programming of parallel axes	■ With X/Y coordinates, independent of machine type; switchover with **FUNCTION PARAXMODE**	■ Machine-dependent with the existing parallel axes
■ Automatic correction of relative references	■ Relative references in contour subprograms are not corrected automatically	■ All relative references are corrected automatically
■ Specify the working plane during programming	■ BLK form ■ **Plane XY ZX YZ** soft key if the working plane differs	■ BLK form
Q-parameter programming:		
■ Q-parameter formula with SGN	Q12 = SGN Q50 ■ if Q 50 = 0 then Q12 = 0 ■ if Q50 > 0 then Q12 = 1 ■ if Q50 < 0 then Q12 = -1	Q12 = SGN Q50 ■ if Q50 >= 0 then Q12 = 1 ■ if Q50 < 0 then Q12 = -1

Function	TNC 640	iTNC 530
Handling of error messages:		
■ Help with error messages	■ Call via **ERR** key	■ Call via **HELP** key
■ Switching the operating mode while help menu is active	■ Help menu is closed when the operating mode is switched	■ Operating mode switchover is not allowed (key is non-functional)
■ Selecting the background operating mode while help menu is active	■ Help menu is closed when F12 is used for switching	■ Help menu remains open when F12 is used for switching
■ Identical error messages	■ Are collected in a list	■ Are displayed only once
■ Acknowledgment of error messages	■ Every error message (even if it is displayed more than once) must be acknowledged, the **DELETE ALL** function is available	■ Error message to be acknowledged only once
■ Access to protocol functions	■ Log and powerful filter functions (errors, keystrokes) are available	■ Complete log without filter functions available
■ Saving service files	■ Available. No service file is generated upon a system crash ■ Error number selectable for which a service file will be generated automatically	■ Available. A service file is generated automatically upon a system crash
Find function:		
■ List of words recently searched for	■ Not available	■ Available
■ Show elements of active block	■ Not available	■ Available
■ Show list of all available NC blocks	■ Not available	■ Available
Starting the search function with the up/down arrow keys when highlighted	Works up to max. 100000 NC blocks, can be set via configu-ration datum	No limitation regarding program length
Programming graphics:		
■ True-to-scale display of grid	■ Available	■ Not available
■ Editing contour subprograms in SLII cycles with **AUTO DRAW ON**	■ With error messages, in the main program the cursor is positioned on the **CYCL CALL** NC block	■ With error messages, the cursor is positioned on the NC block in the contour subprogram that caused the error
■ Moving the zoom window	■ Repeat function not available	■ Repeat function available

Function	TNC 640	iTNC 530
Programming minor axes:		
■ Syntax **FUNCTION PARAXCOMP**: Define the behavior of the display and the paths of traverse	■ Available	■ Not available
■ Syntax **FUNCTION PARAXMODE**: Define the assignment of the parallel axes to be traversed	■ Available	■ Not available
Programming OEM cycles		
■ Access to table data	■ Via **SQL** commands and via **FN 17** and**FN 18** or **TABREAD** / **TABWRITE** functions ■	■ Via **FN 17** and **FN 18** or **TABREAD** / **TABWRITE** functions
■ Access to machine parameters	■ With the **CFGREAD** function	■ Via **FN 18** functions
■ Creating interactive cycles with **CYCLE QUERY**, e.g. touch probe cycles in Manual Operation	■ Available	■ Not available

Comparison: Differences in Test Run, functionality

Function	TNC 640	iTNC 530
Entering a program with the **GOTO** key	Function only possible if the **START SINGLE** soft key was not pressed	Function also possible after **START SINGLE**
Calculation of machining time	Each time the simulation is repeated by pressing the START soft key, the machining time is totaled	Each time the simulation is repeated by pressing the START soft key, time calculation starts at 0
Single block	With point pattern cycles and **CYCL CALL PAT**, the control stops after each point	Point pattern cycles and **CYCL CALL PAT** are handled by the control as a single NC block

Comparison: Differences in Test Run, operation

Function	TNC 640	iTNC 530
Zoom function	Each sectional plane can be selected by individual soft keys	Sectional plane can be selected via three toggle soft keys
Machine-specific miscellaneous functions M	Lead to error messages if they are not integrated in the PLC	Are ignored during Test Run
Displaying/editing the tool table	Function available via soft key	Function not available
Tool depiction	■ Turquoise: Tool length ■ Red: Length of cutting edge and tool is engaged ■ Blue: Length of cutting edge and tool is not engaged	■ - ■ Red: Tool is engaged ■ Green: Tool is not engaged
View options of 3-D view	Available	Function not available
Adjustable model quality	Available	Function not available

Comparison: Differences in programming station

Function	TNC 640	iTNC 530
Demo version	NC programs with more than 100 NC blocks cannot be selected; an error message is issued	NC programs can be selected, max. 100 NC blocks are displayed, further NC blocks are truncated in the display
Demo version	If nesting with **PGM CALL** results in more than 100 NC blocks, there is no test graphic display; an error message is not issued	Nested NC programs can be simulated
Demo version	You can transfer up to 10 elements from the CAD viewer to an NC program.	You can transfer up to 31 lines from the DXF converter to an NC program.
Copying NC programs	Copying to and from the directory **TNC:** is possible with Windows Explorer	**TNCremo** or file manager of programming station must be used for copying
Shifting the horizontal soft-key row	Clicking on the soft-key bar shifts one soft-key row to the right or left	Clicking any soft-key bar activates the respective soft-key row

Index

HEIDENHAIN

DR. JOHANNES HEIDENHAIN GmbH
Dr.-Johannes-Heidenhain-Straße 5
83301 Traunreut, Germany
☏ +49 8669 31-0
FAX +49 8669 32-5061
E-mail: info@heidenhain.de

Technical support FAX +49 8669 32-1000
Measuring systems ☏ +49 8669 31-3104
E-mail: service.ms-support@heidenhain.de
NC support ☏ +49 8669 31-3101
E-mail: service.nc-support@heidenhain.de
NC programming ☏ +49 8669 31-3103
E-mail: service.nc-pgm@heidenhain.de
PLC programming ☏ +49 8669 31-3102
E-mail: service.plc@heidenhain.de
APP programming ☏ +49 8669 31-3106
E-mail: service.app@heidenhain.de

www.heidenhain.de

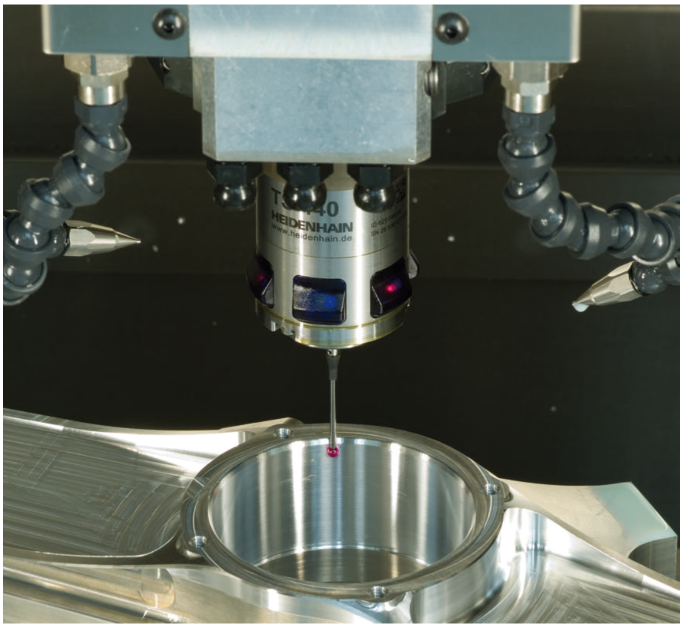
Touch probes from HEIDENHAIN

help you reduce non-productive time and improve the dimensional accuracy of the finished workpieces.

Workpiece touch probes

TS 220 Signal transmission by cable
TS 440 Infrared transmission
TS 642, TS 740 Infrared transmission

- Workpiece alignment
- Preset setting
- Workpiece measurement



Tool touch probes

TT 160 Signal transmission by cable
TT 460 Infrared transmission

- Tool measurement
- wear monitoring
- tool breakage detection

Original document
892903-28 · Ver08 · SW10 · 10/2019 · H · Printed in Germany