HEIDENHAIN

TNC 640

User's Manual
ISO programming

NC Software
340590-09
340591-09
340595-09

English (en)
10/2018

Controls and displays

Keys

If you are using a TNC 640 with touch control, you can replace some keystrokes with hand-to-screen contact.

Further information: "Operating the Touchscreen", Page 473

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

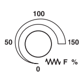
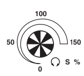
Cycles, subprograms and program section repeats

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	340590-09
TNC 640 E	340591-09
TNC 640 Programming Station	340595-09

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

- Advanced Function Set 2 (option 9) limited to four-axis interpolation
- KinematicsComp (option 52)

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

Display Step (option 23)	
Display step	**Input resolution:** ■ Linear axes down to 0.01 µm ■ Rotary axes to 0.00001°

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 Export license required	Compensation of position and component errors

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

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

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-08

- New **FUNCTION PROG PATH** function for taking the entire tool radius into account in 3-D radius compensation, see "Interpretation of the programmed path", Page 393
- New **FACING HEAD POS** function for working with facing heads, see "Using a facing slide", Page 466
- Touchscreen operation is supported, see "Operating the Touchscreen", Page 473
- When an application is active on the third or fourth desktop, the operating mode keys are also effective with touch operation, see "Save elements and switch to the NC program", Page 484
- Using **DRS** it is now possible to define a cutter radius oversize for a turning tool, see "Tool compensation in the NC program", Page 456
- The **AFC** function (option 45) can now also be used in turning mode, see "Cutting force monitoring with the AFC function", Page 470
- The **M138** function is now also effective in turning mode.
- **CONTOUR DEF** can now also be programmed in ISO format, see "Functions for contour and point machining menu", Page 319
- The **PLANE** functions can now also be programmed in ISO format with **FMAX** and **FAUTO**, see "Specifying the positioning behavior of the PLANE function", Page 372
- New **FUNCTION COUNT** function for controlling a counter, see "Defining a counter", Page 331
- New **FUNCTION LIFTOFF** function for retracting the tool from the contour upon an NC stop, see "Lift off tool at NC stop: FUNCTION LIFTOFF", Page 347
- It is possible to comment out NC blocks, see "Commenting out an existing NC block", Page 184
- The CAD viewer exports points with **FMAX** to an H file, see "Selecting the file type", Page 420
- When multiple instances of the CAD viewer are open, they are shown somewhat smaller on the third desktop.
- The CAD viewer now enables you to extract data from STEP, IGES and STEP files , see "Data Transfer from CAD Files", Page 403
- It is now also possible to transfer undefined Q parameters with the **D00** function.
- With D16, it is possible to enter references to Q parameters or QS parameters as the source and target, see "Basics", Page 275
- The D18 functions have been expanded, see "D18 – Reading system data", Page 282

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

- New **Global Program Settings** function (option 44).
- The new **Batch Process Manager** function enables you to plan production orders.
- New tool-oriented pallet machining function.
- New pallet preset management.

- If a pallet table is selected in a Program Run operating mode, the **Tooling list** and **T usage order** are calculated for the entire pallet table.
- **Dynamic Collision Monitoring (DCM)** is now also available in the **Test Run** operating mode.
- You can also open the tool-carrier files in the file management.
- With the **ADAPT NC PGM / TABLE** function, you can also import and modify freely definable tables.
- The machine tool builder can define update rules that make it possible, for example, to automatically remove umlauts from tables and NC programs when importing a table.
- A quick search for the tool name is possible in the tool table.
- The machine tool builder can disable the setting of presets in individual axes.
- Line 0 of the preset table can also be edited manually.
- The nodes in all tree structures can be expanded and collapsed by double-clicking them.
- New icon in the status display for mirrored machining.
- Graphic settings in the **Test Run** operating mode are permanently stored.
- In the **Test Run** operating mode, you can now choose between various ranges of traverse.
- The tool data of touch probes can also be displayed and entered in the tool management (option 93).
- New MOD dialog for managing radio touch probes.
- With the **TCH PROBE MONITOR OFF** soft key you can suppress touch-probe monitoring for 30 seconds.
- During manual probing **ROT** and **P**, workpiece misalignment can be compensated by aligning a rotary table.
- If the function for orienting the touch probe to the programmed probe direction is active, the number of spindle revolutions is limited when the guard door is open. In some cases, the direction of spindle rotation will change so that positioning will not always follow the shortest path.
- New machine parameter **iconPrioList** (no. 100813) for defining the order of icons in the status display.
- New machine parameter **suppressResMatlWar** (no. 201010) for suppressing the **Remaining material** warning.
- The machine parameter **clearPathAtBlk** (no. 124203) enables you to specify whether the tool paths will be cleared with a new BLK FORM in the **Test Run** operating mode.
- New optional machine parameter **CfgDisplayCoordSys** (no. 127500) for selecting the coordinate system in which a datum shift is to be shown in the status display.
- The control now supports up to 24 control loops, including a maximum of four spindles.

Modified functions 34059x-08

- If you use locked tools, the control displays a warning in the **Programming** operating mode, see "Programming graphics", Page 198
- The **M94** miscellaneous function is effective for all rotary axes that are not limited by software limit switches or traverse limits, see "Reducing display of a rotary axis to a value less than 360°: M94", Page 386
- Holes and threads are shown in light blue in the programming graphics, see "Programming graphics", Page 198
- The sort order and the column widths are retained in the tool selection window when the control is switched off, see "Calling the tool data", Page 122
- If a subprogram called with %:PGM ends with **M2** or **M30**, the control issues a warning. The control automatically clears the warning as soon as you select another NC program, see "Programming notes", Page 242
- The time needed to paste a large amount of data into an NC program was considerably reduced.
- When you double-click a selection field of the table editor with the mouse or press the **ENT** key, a pop-up window opens.
- The machine tool builder configures whether the control will take the axis angle into account or set it to 0 for the axes specified in **M138**, see "Selecting tilting axes: M138", Page 390
- The **SYSSTR** function can be used to read the path of pallet programs, see "Reading system data", Page 295
- A programmed limitation of the spindle speed is restored after eccentric turning.

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

- If you use locked tools, the control displays a warning in the **Test Run** operating mode.
- The control provides a positioning logic for returning to the contour.
- The positioning logic for returning to the contour with a replacement tool has changed.
- Axes that are not active in the current kinematic model can also be referenced in a tilted working plane.
- The tool is shown in red in the graphics while it is in contact with the workpiece, and blue during air cuts.
- The positions of the sectional planes are no longer reset when a program or a new blank form is selected.
- Spindle speeds can be entered with decimal places also in the **Manual operation** mode. The control displays the decimal places when the spindle speed is < 1000.
- The control displays an error message in the header until it is cleared or replaced by a higher-priority error.
- To connect a USB stick you no longer have to press a soft key.
- The speed of setting the jog increment, spindle speed and feed rate was adjusted for electronic handwheels.
- The icons of basic rotation, 3-D basic rotation and tilted working plane were modified to make them easier to distinguish.

- The icon for **FUNCTION TCPM** was modified.
- The icon for the **AFC** function was modified.
- The control automatically recognizes whether a table is to be imported or the table format is to be adapted.
- If no AFC table with cutting data is available yet, the control opens an empty AFC table when the **AFC SETTINGS** soft key is pressed.
- When you place the cursor in an input field of the tool management, the entire input field is highlighted.
- When configuration subfiles are modified, the control no longer aborts the test run, but only displays a warning.
- You can neither set nor modify a preset without having referenced the axes.
- The control issues a warning if the handwheel potentiometers are still active when the handwheel is deactivated.
- When using the HR 550 or HR 550FS handwheel, a warning is issued if the battery voltage is too low.
- The machine tool builder can define whether the **R-OFFS** offset will be taken into account for a tool with **CUT** 0.
- The machine tool builder can change the simulated tool change position.
- When saving the live image, you can select the target directory and the file name.
- In the machine parameter **decimalCharakter** (no. 100805) you can define whether a period or a comma will be used as the decimal separator.

New and modified cycle functions 34059x-08

Further information:Cycle Programming User's Manual

- New Cycle 453 **KINEMATICS GRID**. This cycle makes it possible to probe a calibration sphere in multiple tilting-axis positions predefined by the OEM. The measured deviations can be compensated via compensation tables. Options 48 **KinematicsOpt** and 52 **KinematicsComp** are required; the machine tool builder has to adapt the feature to the respective machine.
- New Cycle 441 **FAST PROBING**. With this cycle you can set various touch probe parameters (e.g. positioning feed rate) that are globally effective for all subsequently used touch probe cycles.
- The parameters Q215, Q385, Q369 and Q386 were added to Cycles 256 **RECTANGULAR STUD** and 257 **CIRCULAR STUD**.
- The recessing cycles 860 to 862 and 870 to 872 were extended by the input parameter Q211. In this parameter, a dwell time can be specified in revolutions of the workpiece spindle, which retards the retraction after the recessing on the floor.
- Cycle 239 ascertains the current load of the machine axes with the LAC control function. In addition, Cycle 239 can now also adjust the maximum axis acceleration. Cycle 239 supports the determination of the load on synchronized axes.
- The feed rate behavior in Cycles 205 and 241 was changed.
- Changes of details in Cycle 233: Monitors the tooth length (**LCUTS**) during finishing, increases the area by Q357 in the milling direction when roughing with milling strategies 0 to 3 (provided that no limit has been set in the milling direction).
- **CONTOUR DEF** can be programmed in ISO format.
- The technologically outdated Cycles 1, 2, 3, 4, 5, 17, 212, 213, 214, 215, 210, 211, 230, and 231 grouped under **OLD CYCLES** can no longer be inserted using the editor. These cycles can still be executed and edited, however.
- The tool touch probe cycles, such as Cycles 480, 481 and 482, can be hidden.
- Cycle 225 Engraving can engrave the current counter reading by using a new syntax.
- New SERIAL column in the touch probe table.
- Enhancement of the contour train: Cycle 25 with Residual Material Machining, Cycle 276 3-D Contour Train.

New functions 34059x-09

- It is now possible to work with cutting data tables, see "Working with cutting data tables", Page 195
- New **PLANE XY ZX YZ** soft key for selecting the working plane during FK programming, see "Fundamentals", Page 167
- In **Test Run** operating mode, a counter defined in the NC program is simulated, see "Defining a counter", Page 331
- 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 403

- You can now use QS parameters to read from and write to freely definable tables, see "D27 – Write to a freely definable table", Page 340
- The D16 function was expanded to include the ***** input character that can be used to write comment lines, see "Creating a text file", Page 275
- New output format for the D16 function **%RS** that you can use to output texts without formatting, see "Creating a text file", Page 275
- The D18 functions have been expanded, see "D18 – Reading system data", Page 282

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 turn command over 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 selection option **SYM** in addition to **SEQ**, see "Specifying the positioning behavior of the PLANE function", Page 372
- The cutting data calculator has been improved, see "Cutting data calculator", Page 193
- The **CAD-Viewer** now outputs **PLANE SPATIAL** instead of **PLANE VECTOR**, see "Defining the datum", Page 413
- The **CAD-Viewer** now outputs 2-D contours by default.
- The control does not run a tool change macro if neither a tool name nor a tool number is programmed in the tool call, but the same tool axis as in the previous **T** block, see "Calling the tool data", Page 122
- The control issues an error message if you combine an FK block with M89.
- When using the D16 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 281

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**.
- 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 prepositioning.
- 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 also uses Q207 FEED RATE FOR MILLNG for roughing.
- 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.

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:

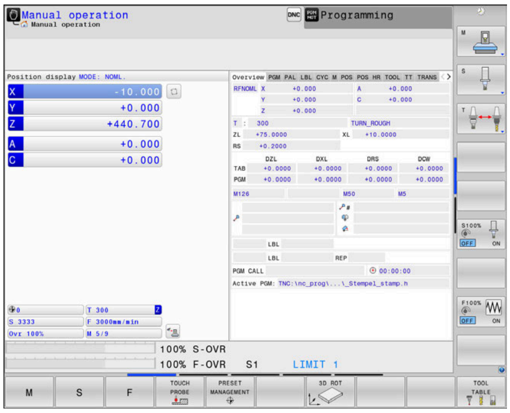
- Switching on the machine
- Programming the workpiece

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.

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

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

Select operating mode

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

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

Further information on this topic

- Operating modes
  Further information: "Programming", Page 68

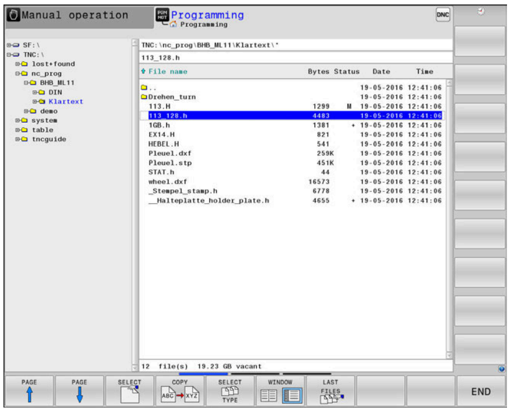
Important controls and displays

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 94
- Overview of keys
  Further information: "Controls and displays", Page 2

Creating a new NC program / file management



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.

- Use the arrow keys to select the folder in which you want to open the new file
- Enter any desired file name with the extension **.i**

ENT

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

MM

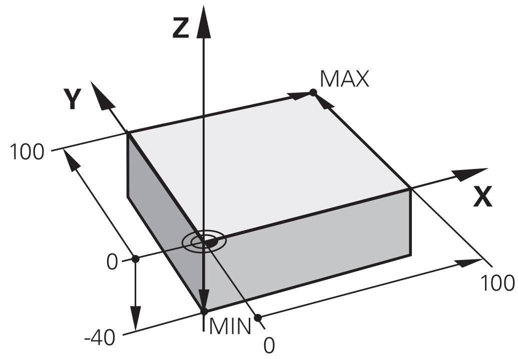
- Select the unit of measure: Press the **MM** or **INCH** soft key

The control automatically generates the first and last NC blocks of the NC program. Afterwards you can no longer change these NC blocks.

Further information on this topic

- File management
  Further information: "File management", Page 100
- Creating a new NC program
  Further information: "Opening and entering NC programs", Page 86

Defining a workpiece blank



After you have created a new NC program you can define a workpiece blank. For example, define a cuboid by entering the MIN and MAX points, each with reference to the selected preset.

After you have selected the desired blank form via soft key, the control automatically initiates the workpiece blank definition and asks for the required data:

- **Spindle axis Z - Plane XY**: Enter the active spindle axis. G17 is saved as default setting. Accept with the **ENT** key
- **Workpiece blank def.: Minimum X**: Enter the smallest X coordinate of the workpiece blank with respect to the preset, e.g. 0, confirm with the **ENT** key
- **Workpiece blank def.: Minimum Y**: Enter the smallest Y coordinate of the workpiece blank with respect to the preset, e.g. 0, confirm with the **ENT** key
- **Workpiece blank def.: Minimum Z**: Enter the smallest Z coordinate of the workpiece blank with respect to the preset, e.g. -40, confirm with the **ENT** key
- **Workpiece blank def.: Maximum X**: Enter the largest X coordinate of the workpiece blank with respect to the preset, e.g. 100, confirm with the **ENT** key
- **Workpiece blank def.: Maximum Y**: Enter the largest Y coordinate of the workpiece blank with respect to the preset, e.g. 100, confirm with the **ENT** key
- **Workpiece blank def.: Maximum Z**: Enter the largest Z coordinate of the workpiece blank with respect to the preset, e.g. 0, confirm with the **ENT** key

> The control ends the dialog.

Example

```
%NEW G71 *
N10 G30  G17 X+0  Y+0 Z-40*
N20 G31  X+100 Y+100  Z+0*
N99999999 %NEW  G71 *
```

Further information on this topic

- Define workpiece blank
  Further information: "Creating a new NC program",
  Page 90

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

```
%BSPCONT G71 *
N10 G30  G71 X...  Y... Z...*
N20 G31  X... Y...  Z...*
N30 T5  G17 S5000*
N40 G00  G40 G90  Z+250*
N50 X...  Y...*
N60 G01  Z+10 F3000  M13*
N70 X...  Y... RL  F500*
...
N160 G40  ... X...  Y... F3000  M9*
N170 G00  Z+250 M2*
N99999999 BSPCONT  G71 *
```

1 Call tool, define tool axis
2 Retract the tool
3 Pre-position the tool in the working plane near the contour starting point
4 In the tool axis, position the tool above the workpiece, or preposition immediately to workpiece depth. If required, switch on the spindle/coolant
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 workpiece machining", Page 134

Recommended program layout for simple cycle programs

Example

```
%BSBCYC G71 *
N10 G30  G71 X...  Y... Z...*
N20 G31  X... Y...  Z..*
N30 T5  G17 S5000*
N40 G00  G40 G90  Z+250*
N50 G200...*
N60 X...  Y...*
N70 G79  M13*
N80 G00  Z+250 M2*
N99999999 BSBCYC  G71 *
```

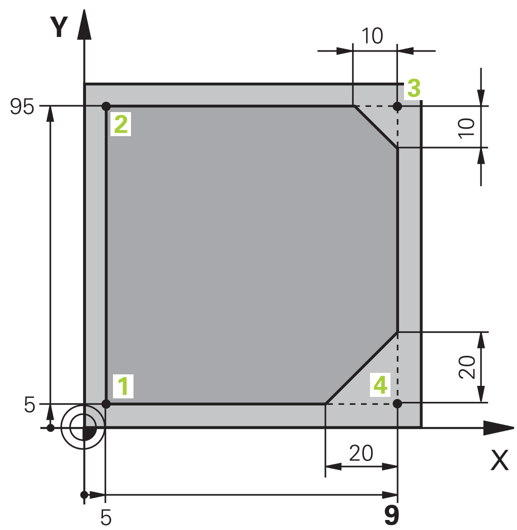
1 Call tool, define tool axis
2 Retract the tool
3 Define the fixed cycle
4 Move to the machining position
5 Call the cycle, switch on the spindle/coolant
6 Retract the tool, end the NC program

Further information on this topic

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

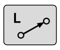
Programming a simple contour

The contour shown to the right is to be milled once to a depth of 5 mm. You have already defined the workpiece blank. After you have initiated a dialog through a function key, enter all the data requested by the control in the screen header.



- Call the tool: Enter the tool data. Confirm the entry in each case with the **ENT** key, and do not forget the **G17** tool axis
- Press the **L** key to open a program block for a linear movement
- Press the left arrow key to switch to the input range for G codes
- Press the **G00** soft key if you want to enter a rapid traverse motion
- Press the **G90** soft key for absolute values
- Retract tool: Press the orange axis key **Z** and enter the value for the position to be approached, e.g. 250. Press the **ENT** key
- Activate no radius compensation: Press the **G40** soft key
- Confirm **Miscellaneous function M?** with the **END** key
- > The control stores the entered positioning block.
- Press the **L** key to open an NC block for a linear movement
- Press the left arrow key to switch to the input range for G codes
- Press the **G00** soft key if you want to enter a rapid traverse motion
- Preposition the tool in the working plane: Press the orange **X** axis key and enter the value for the position to be approached, e.g. –20
- Press the orange axis key **Y** and enter the value for the position to be approached, e.g. -20. Confirm your entry with the ENT key.
- Activate no radius compensation: Press the **G40** soft key
- Confirm **Miscellaneous function M?** with the **END** key
- > The control stores the entered positioning block.
- Press the **L** key to open an NC block for a linear movement
- Press the left arrow key to switch to the input range for G codes
- Press the **G00** soft key if you want to enter a rapid traverse motion

- Move tool to working depth: Press the orange axis key **Z** and enter the value for the position to be approached, e.g. -5. Press the **ENT** key
- Activate no radius compensation: Press the **G40** soft key
- **Miscellaneous function M?** Switch on the spindle and coolant, e.g. **M13**, and confirm with the **END** key
- > The control stores the entered positioning block.
- Press the **L** key to open an NC block for a linear movement
- Enter the coordinates of the contour starting point **1** in X and Y, e.g. 5/5, and confirm with the **ENT** key
- Activate radius compensation to the left of the path: Press the **G41** soft key
- **Feed rate F=?** Enter the machining feed rate, e.g. 700 mm/min, save your entry with the **END** key
- Enter **26** to approach the contour: Define **Rounding-off radius?** for the circular arc, save entries with the **END** key
- Machine the contour and move to contour point **2**: You only need to enter the information that changes. In other words, enter only the Y coordinate 95 and save your entry with the **END** key
- Move to contour point **3**: Enter the X coordinate 95 and save your entry with the **END** key
- Define chamfer **G24** on contour point **3**: **Chamfer side length?** Enter 10 mm, save with the **END** key
- Move to contour point **4**: Enter the Y coordinate 5 and save your entry with the **END** key
- Define chamfer **G24** on contour point **4**: **Chamfer side length?** Enter 20 mm, save with the **END** key
- Move to contour point **1**: Enter the X coordinate 5 and save your entry with the **END** key
- Enter **27** to depart from the contour: Define the **Rounding-off radius?** of the departing arc
- Depart contour: Enter coordinates outside of the workpiece in X and Y, e.g. -20/-20, confirm with the **ENT** key
- Activate no radius compensation: Press the **G40** soft key



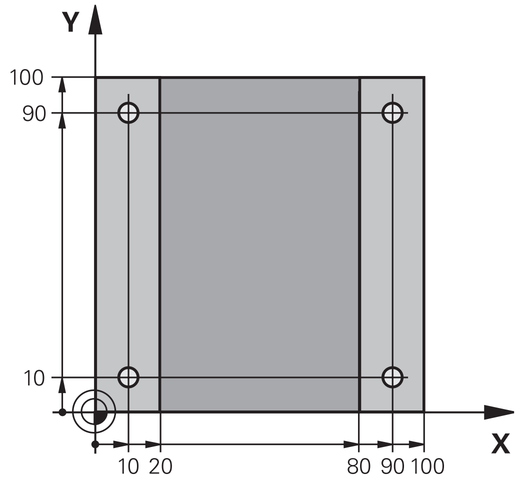
- Press the **L** key to open an NC block for a linear movement
- Press the **G00** soft key if you want to enter a rapid traverse motion
- Retract tool: Press the orange axis key **Z** to retract in the tool axis, and enter the value for the position to be approached, e.g. 250. Press the **ENT** key
- Activate no radius compensation: Press the **G40** soft key
- **Miscellaneous function M?** Enter **M2** to end the program, then confirm with the **END** key
- > The control stores the entered positioning block.

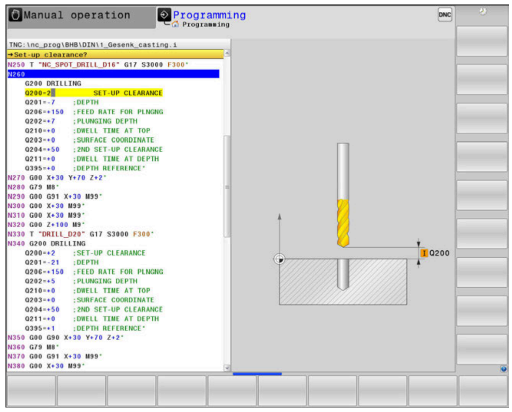
Further information on this topic

- Complete example with NC blocks
  Further information: "Example: Linear movements and chamfers with Cartesian coordinates", Page 157
- Creating a new NC program
  Further information: "Opening and entering NC programs", Page 86
- Approaching/departing contours
  Further information: "Approaching and departing a contour", Page 137
- Programming contours
  Further information: "Overview of path functions", Page 148
- Tool radius compensation
  Further information: "Tool radius compensation ", Page 128
- Miscellaneous functions M
  Further information: "Miscellaneous functions for program run inspection, spindle and coolant ", Page 216

Creating a cycle program

The holes (depth of 20 mm) shown in the figure at right are to be drilled with a standard drilling cycle. You have already defined the workpiece blank.





TOOL CALL

- Call the tool: Enter the tool data. Confirm the entry in each case with the **ENT** key, do not forget the tool axis

L

- Press the **L** key to open an NC block for a linear movement
- Press the left arrow key to switch to the input range for G codes

G00

- Press the **G00** soft key if you want to enter a rapid traverse motion
- Press the **G90** soft key for absolute values
- Retract tool: Press the orange axis key **Z** and enter the value for the position to be approached, e.g. 250. Press the **ENT** key
- Activate no radius compensation: Press the **G40** soft key
- **Miscellaneous function M?** Switch on the spindle and coolant, e.g. **M13**,and confirm with the **END** key
- \> The control stores the entered positioning block.

CYCL DEF

- Call the cycle menu: Press the **CYCL DEF** key

DRILLING/ THREAD

- Display the drilling cycles

200

- Select standard drilling cycle 200
- \> The control starts the dialog for cycle definition.
- Enter all parameters requested by the control step by step and conclude each entry with the **ENT** key
- \> In the screen to the right, the control also displays a graphic showing the respective cycle parameter

G

- Enter **0** to approach the first drilling position: Enter the **coordinates** of the drilling position, call the cycle with **M99**

G

- Enter **0** to move to further drilling positions: Enter the **coordinates** of the specific drilling positions, and call the cycle with **M99**

G

- Enter **0** to retract the tool: Press the orange axis key **Z** and enter the value for the position to be approached, e.g. 250. Press the **ENT** key
- **Miscellaneous function M?** Enter **M2** to end the program, then confirm with the **END** key
- \> The control stores the entered positioning block.

Example

%C200 G71 *	
N10 G30 G17 X+0 Y+0 Z-40*	Workpiece blank definition
N20 G31 X+100 Y+100 Z+0*	
N30 T5 G17 S4500*	Tool call
N40 G00 G90 Z+250 G40*	Retract the tool
N50 G200 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	
N60 G00 X+10 Y+10 M13 M99*	Spindle and coolant on, call the cycle
N70 G00 X+10 Y+90 M99*	Call the cycle
N80 G00 X+90 Y+10 M99*	Call the cycle
N90 G00 X+90 Y+90 M99*	Call the cycle
N100 G00 Z+250 M2*	Retract the tool, end program
N99999999 %C200 G71 *	

Further information on this topic

- Creating a new NC program
  Further information: "Opening and entering NC programs", Page 86
- 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 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 or DNC mode.

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

Compatibility

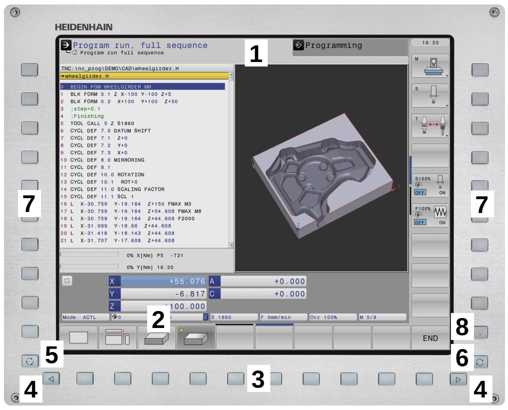
NC programs created on HEIDENHAIN contouring controls (starting from the TNC 150 B) may not always run on the TNC 640. If the NC blocks contain invalid elements, the control will mark these as ERROR blocks or with error messages when the file is opened.

> 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 526

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

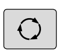
8 Keys for switching the soft keys for machine tool builders

> If you are using a TNC 640 with touch control, you can replace some keystrokes with hand-to-screen contact.
> **Further information:** "Operating the Touchscreen", Page 473

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:



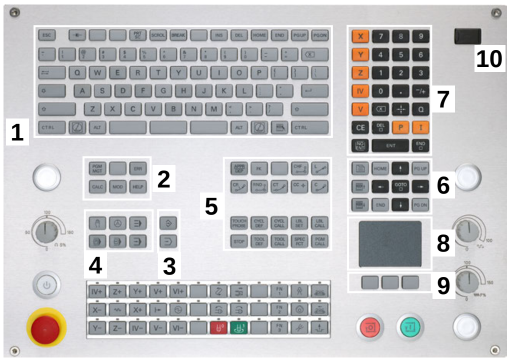
- Press the **screen layout** key: The soft-key row shows the available layout options
  Further information: "Modes of operation", Page 67

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:



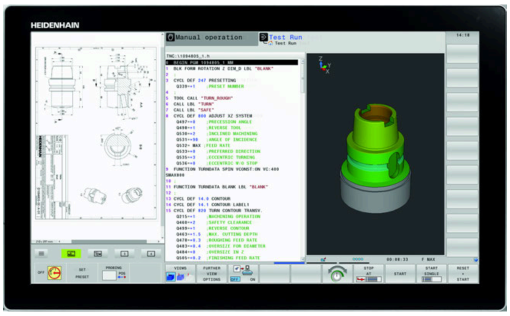
1. Alphabetic keyboard for entering texts and file names, as well as for ISO programming
2.
   - File management
   - Calculator
   - MOD function
   - HELP function
   - Show error messages
   - Toggle between the operating modes
3. Programming modes
4. Machine operating modes
5. Initiating programming 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 hand-to-screen contact.
>
> **Further information:** "Operating the Touchscreen", Page 473

> 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.

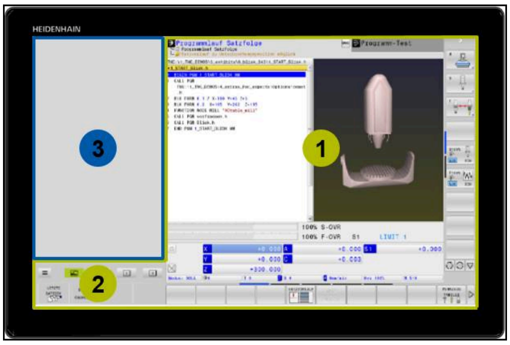
The additional screen workspace in **Extended Workspace Compact** mode provides full multitouch support. When you switch to full-screen mode, you can use the HEIDENHAIN keyboard for your external applications.

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.

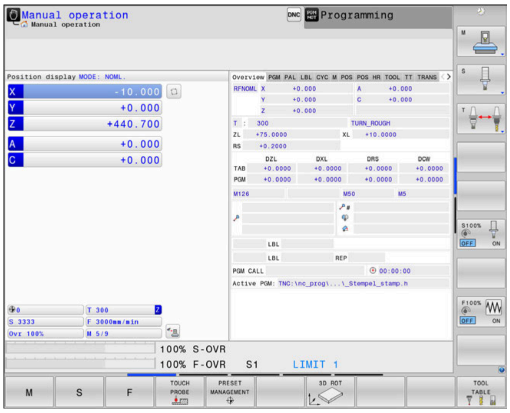
Contents of the **OEM** area:

- 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



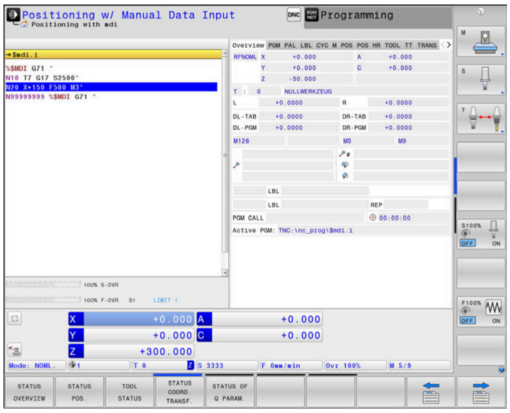
The **Manual operation** mode is required for setting up the machine tool. In this mode of operation, you can position the machine axes manually or by increments, set the presets and tilt the working plane.

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



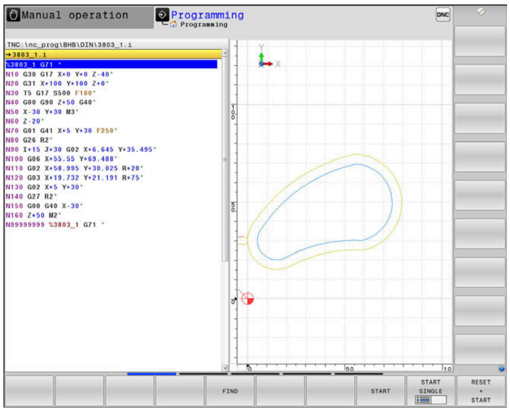
This mode of operation is used for programming simple traversing movements, such as for face milling or pre-positioning.

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.

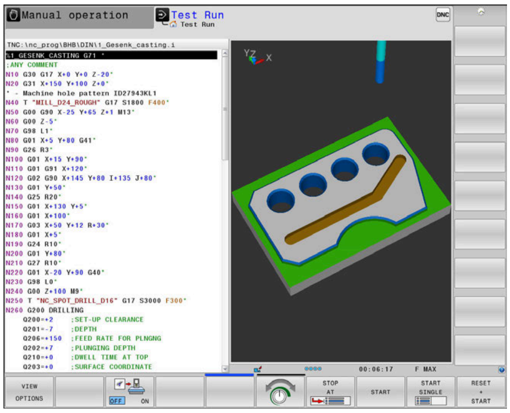


Soft keys for selecting the screen layout

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

Test Run

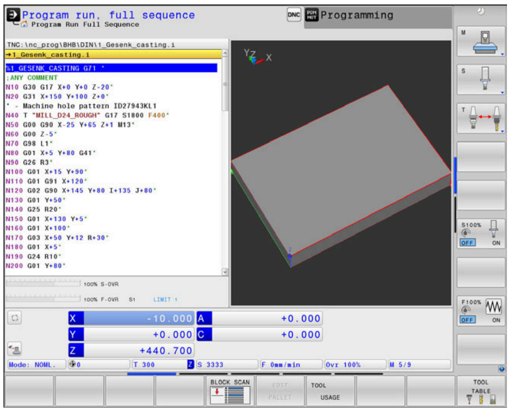
In the **Test Run** operating mode, the control checks NC programs and program sections for errors, such as geometrical incompatibilities, missing or incorrect data within the NC program or violations of the working space. This simulation is supported graphically in different display modes.



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

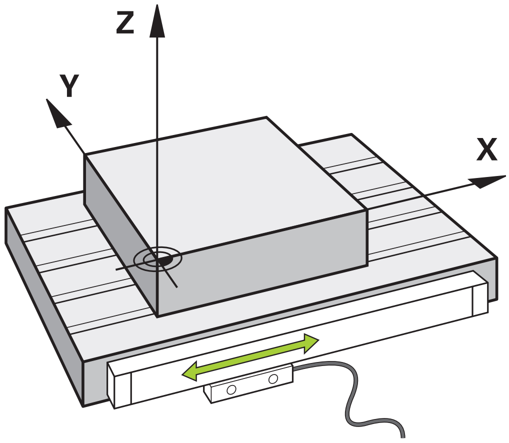
Soft key	Window
PGM	NC program
PROGRAM + SECTS	Left: NC program, right: structure
PROGRAM + STATUS	Left: NC program, right: status display
PROGRAM + WORKPIECE	Left: NC program, right: workpiece
WORKPIECE	Workpiece
POSITION + MACHINE	Left: NC program, right: collision objects and workpiece
MACHINE	Collision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

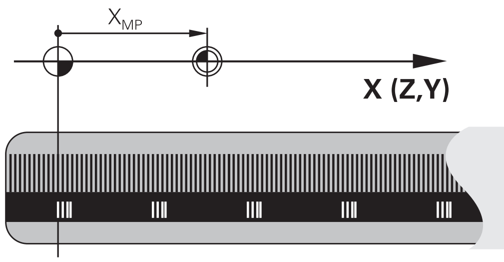
Soft keys for screen layout with pallet tables

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.

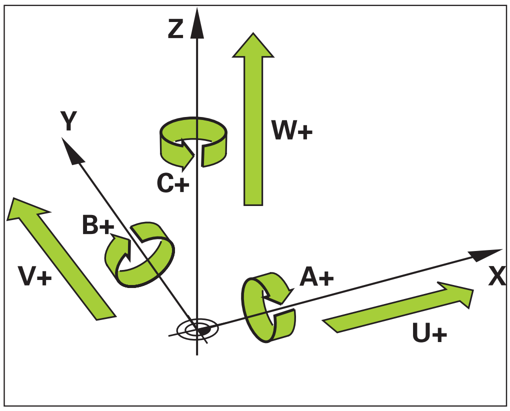
If there is a power interruption, the calculated position will no longer correspond to the actual position of the machine slide. To recover this assignment, incremental position encoders are provided with reference marks. When a reference mark is crossed over, a signal identifying a machine-based reference 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.

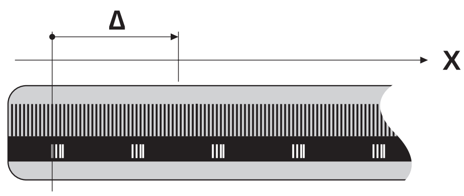
The number, designation and assignment of the programmable axes depend on the machine.

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

Reference systems

For the control to traverse an axis according to a defined path it requires a **reference system**.

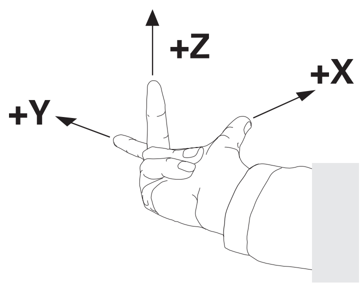
A paraxially mounted linear encoder on a machine tool serves as a simple reference system for linear axes. The linear encoder represents a **number ray**, a unidimensional coordinate system.



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

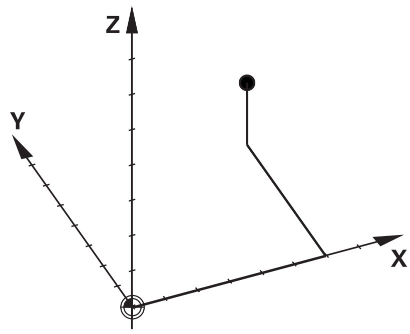
To approach a point in the **space**, the control requires three axes and therefore a reference system with three dimensions. If these three axes are configured perpendicular to each other this creates a so-called **three-dimensional Cartesian coordinate system**.

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



For a point to be uniquely determined in space, a **coordinate origin** is needed in addition to the configuration 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**.

The control must differentiate between various reference systems for it to always perform a tool change at the same position for example, or carry out a machining operation always related to the current workpiece position.



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

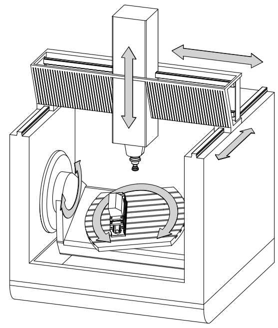
The machine coordinate system corresponds to the description of kinematics and therefore to the actual mechanical design of the machine tool.

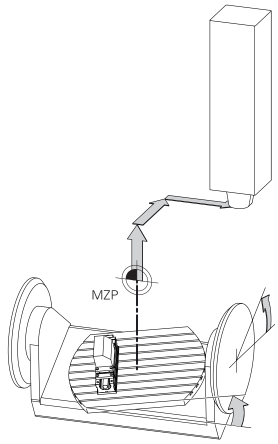
Because the mechanics of a machine tool never precisely correspond to a Cartesian coordinate system, the machine coordinate system consists of several one-dimensional coordinate systems. These one-dimensional coordinate systems correspond to the physical machine axes that are not necessarily perpendicular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

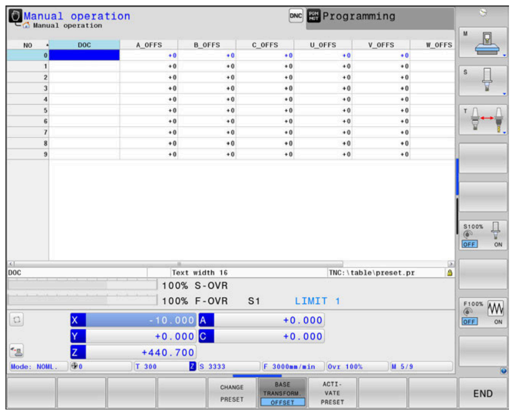
Because the machine configuration values cannot be modified by the user, the machine coordinate system is used for determining constant positions, e.g. the tool change point.





Machine datum (MZP)

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



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

Further information: 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) result in the difference between the **ACTL.** position and the **RFACTL** position of an axis.

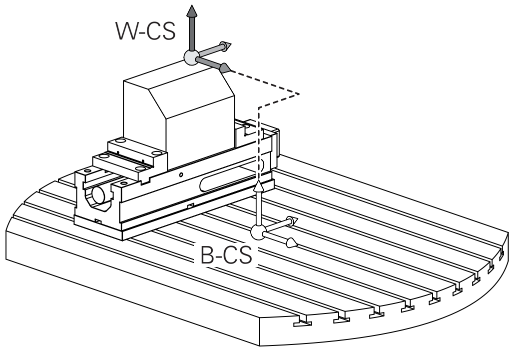
The control converts all movements in the machine coordinate system, independent of the reference system used for value input.

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

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

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

Basic coordinate system B-CS

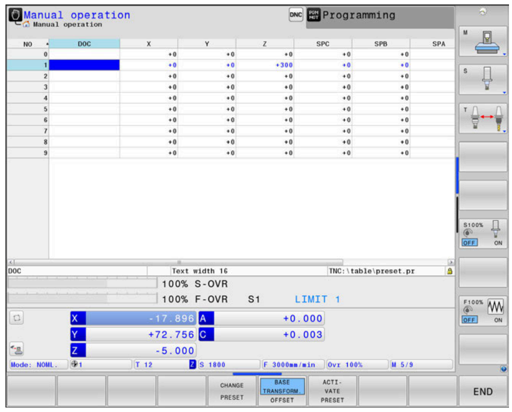


The basic coordinate system is a 3-D Cartesian coordinate system. Its coordinate origin is the end of the kinematics model.

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

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

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



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

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

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

NOTICE

Danger of collision!

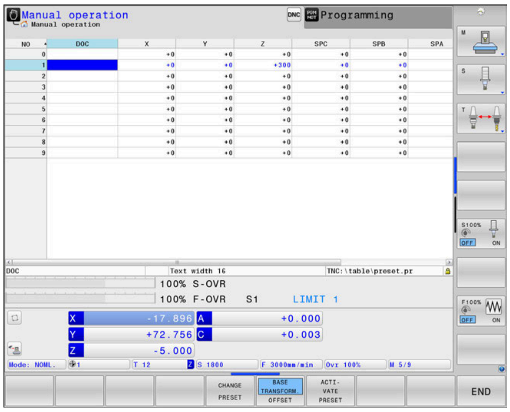
Your control may feature an additional pallet preset table, depending on the machine. In this table the machine tool builder can define **BASE TRANSFORM.** values that take effect before the **BASE TRANSFORM.** values you specify in the preset table become effective. The **PAL** tab of the additional 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!

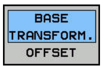
- Refer to the machine tool builder's documentation
- Use pallet presets only in conjunction with pallets
- Check the display of the **PAL** tab before you start machining

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.



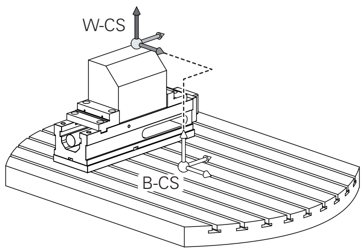
Soft key	Application
	The user determines the position and orientation of the workpiece coordinate system by using a 3-D touch probe for example. The control saves the values determined with respect to the basic coordinate system as **BASE TRANSFORM.** values in the preset management.

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

The **Global Program Settings** function (option 44) additionally provides the following transformations:

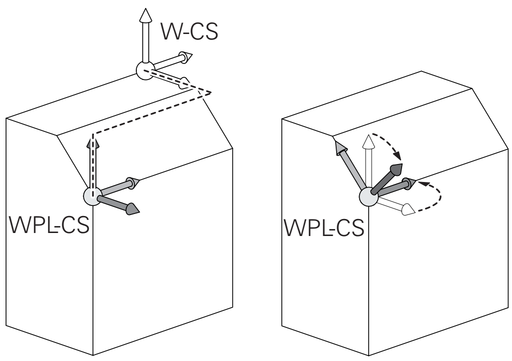
- The **Additive basic rotat. (W-CS)** is added to a basic rotation or a 3-D basic rotation from the preset table and the pallet preset table. The **Additive basic rotat. (W-CS)** is the first transformation that is possible in the workpiece coordinate system (W-CS).
- **Shift (W-CS)** is added to the 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)



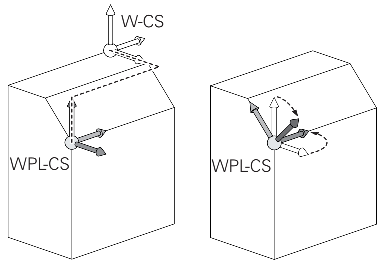
> The result of transformations built up on each other depends on the programming sequence.
>
> In every coordinate system, program only the specified (recommended) transformations. This applies to both setting and resetting the transformations. Any other use may lead to unexpected or undesired results. 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 78

Working plane coordinate system WPL-CS

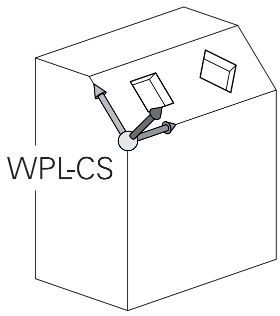
The working plane coordinate system is a 3-D Cartesian coordinate system.

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

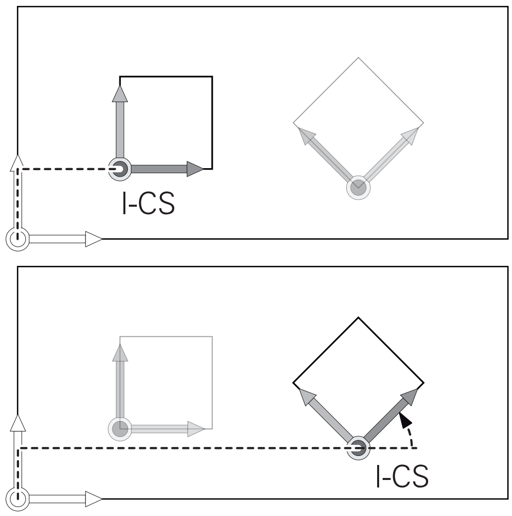


> Without active transformations in the workpiece coordinate system, the position and orientation of the working plane coordinate system and workpiece coordinate system are identical.
>
> There are no transformations in the workpiece coordinate system on 3-axis machine tools or with pure 3-axis machining. 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.

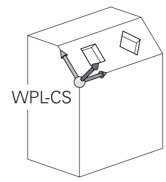
Without active transformations in the working plane coordinate system, the position and orientation of the input coordinate system and working plane coordinate system are identical.

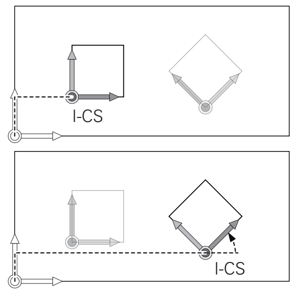
There are also no transformations in the workpiece coordinate system on 3-axis machine tools or with pure 3-axis machining. The **BASE TRANSFORM.** values of the active line of the preset table have a direct effect on the input coordinate system with this assumption.

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.
>
> There are also 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.

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

Positioning blocks in input coordinate system:

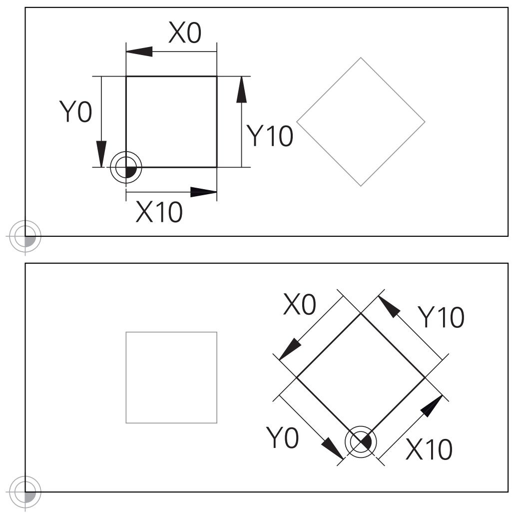
- Paraxial positioning blocks
- Positioning blocks with Cartesian or polar coordinates

Example

```
N70  X+48 R+*
```

```
N70  G01 X+48 Y+102 Z-1.5 R0*
```

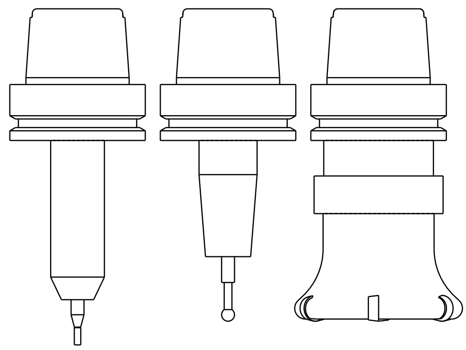
> Orientation of the tool coordinate system can be performed in various reference systems.
>
> **Further information:** "Tool coordinate system T-CS", Page 81



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

Tool coordinate system T-CS

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

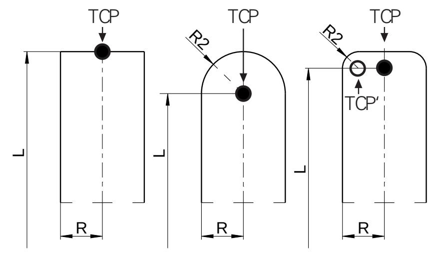


Further information: User's Manual for Setup, Testing and Running NC Programs

> 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.

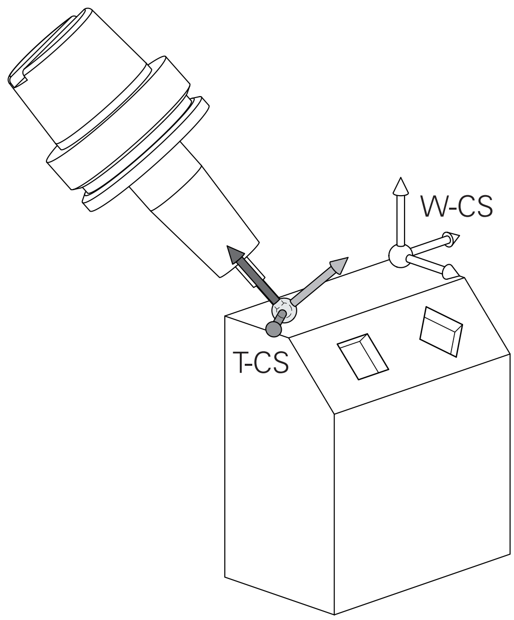


> The position of the TCP as shown in the diagram is obligatory in conjunction with the 3-D tool compensation.

> 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.

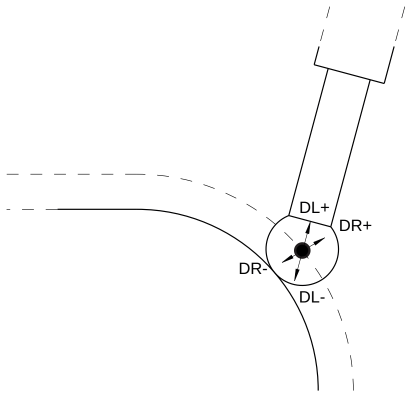
If miscellaneous function **M128** is active, the orientation of the tool coordinate system depends on the tool's current angle of inclination.

Tool angle of inclination in the machine coordinate system:



Example

```
N70 G01 X+10 Y+45 A+10 C+5 R0 M128*
```



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

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

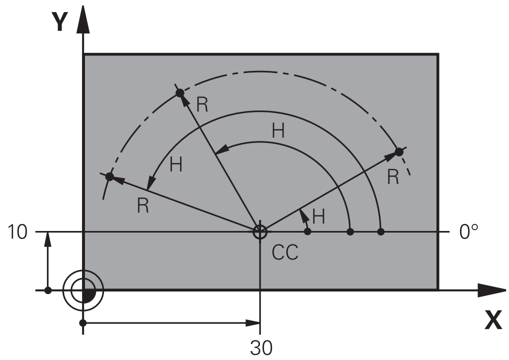
Without the **TCPM** function or miscellaneous function **M128**, orientation of the tool coordinate system and input coordinate system is identical.

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

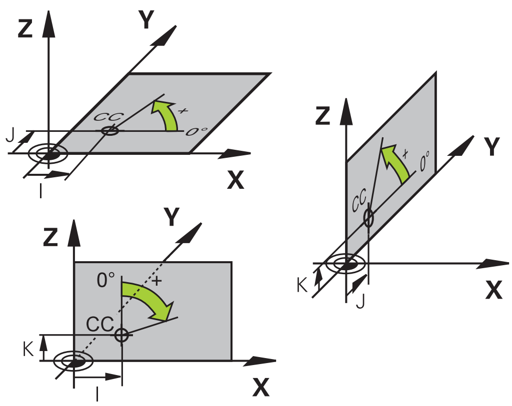


If the production drawing is dimensioned in Cartesian coordinates, you write the NC program using Cartesian coordinates. For parts containing circular arcs or angles, it is often simpler to give the dimensions in polar coordinates.

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 H.

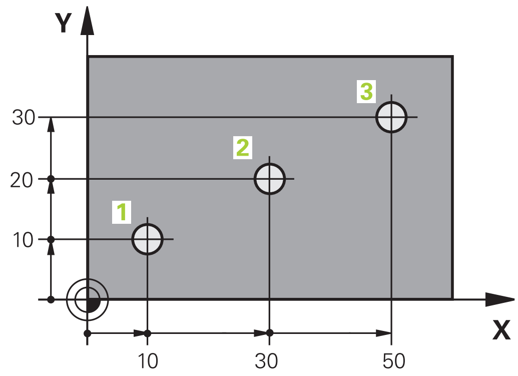
Coordinates of the pole (plane)	Angle reference axis
X/Y	+X
Y/Z	+Y
Z/X	+Z

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



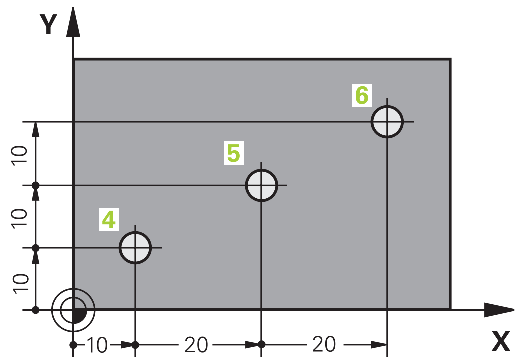
Hole **1**	Hole **2**	Hole **3**
X = 10 mm	X = 30 mm	X = 50 mm
Y = 10 mm	Y = 20 mm	Y = 30 mm

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 G91 function before the axis.

Example 2: Holes dimensioned in incremental coordinates



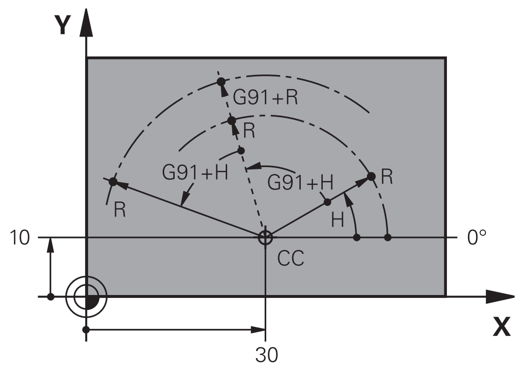
Absolute coordinates of hole **4**
X = 10 mm
Y = 10 mm

Hole **5**, with respect to **4**	Hole **6**, with respect to **5**
G91 X = 20 mm	G91 X = 20 mm
G91 Y = 10 mm	G91 Y = 10 mm

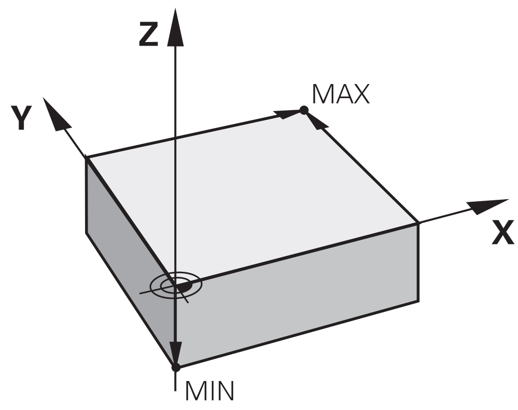
Absolute and incremental polar coordinates

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 datum



A production drawing identifies a certain form element of the workpiece, usually a corner, as the absolute preset (datum). When setting the preset, you first align the workpiece along the machine axes, and then move the tool in each axis to a defined position relative to the workpiece. Set the display of the control either to zero or to a known position value for each position. This establishes the reference system for the workpiece used for the control's display or your NC program.

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

Further information: Cycle Programming User's Manual

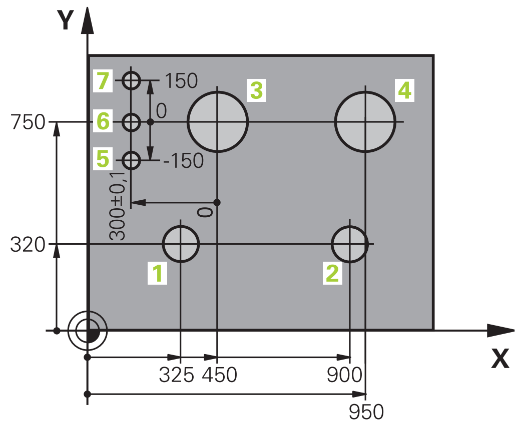
If the production drawing is not dimensioned for NC, set the preset at a position or corner on the workpiece from which the dimensions of the remaining workpiece positions can be measured.

The fastest, easiest and most accurate way of presetting is by using 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 with respect to an absolute preset with the coordinates X=0 Y=0. The coordinates of holes **5** to **7** refer to the relative datum with the absolute coordinates X=450 Y=750. By using the **Datum shift** cycle you can shift the datum temporarily to the position X=450, Y=750 and program the holes (**5** to **7**) without further calculations.



3.5 Opening and entering NC programs

Structure of an NC program in ISO format

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

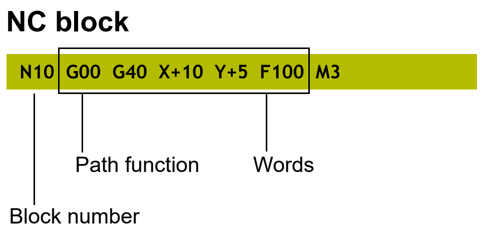
The control numbers the NC blocks of an NC program automatically, depending on the setting in the machine parameter **blockIncrement** (105409). The **blockIncrement** machine parameter (105409) defines the block number increment.

The first NC block of an NC program is identified by **%**, 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 **N99999999**, the program name, and the active unit of measure.



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!

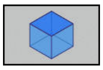
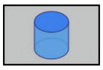
- ▶ If necessary, program an additional safe auxiliary position

Defining the blank: G30/G31

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

> 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 G30: the smallest X, Y and Z coordinates of the blank form, entered as absolute values.
- MAX point G31: the largest X, Y and Z coordinates of the blank form, entered as absolute or incremental values

Example

%NEW G71 *	Program begin, name, unit of measure
N10 G30 G17 X+0 Y+0 Z-40*	Spindle axis, MIN point coordinates
N20 G31 X+100 Y+100 Z+0*	MAX point coordinates
N99999999 %NEW G71 *	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

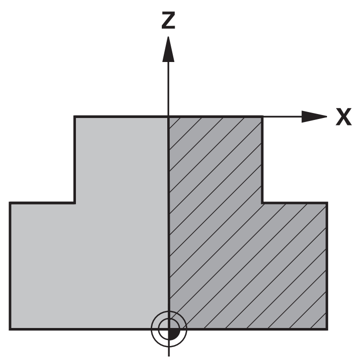
%NEW G71 *	Program begin, name, unit of measure
N10 BLK FORM CYLINDER Z R50 L105 DIST+5 RI10*	Spindle axis, radius, length, distance, inside radius
N99999999 %NEW G71 *	Program end, name, unit of measure

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.

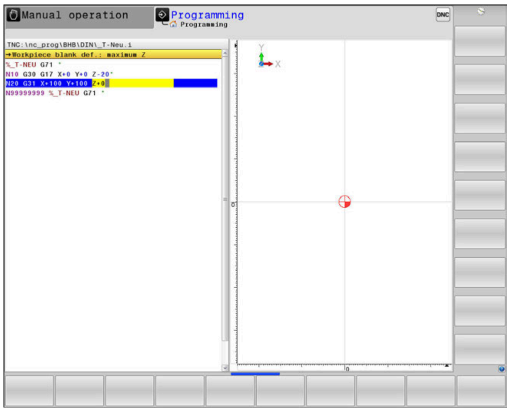
If you define a rotationally symmetric blank with incremental coordinates, the dimensions are then independent of the diameter programming.

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

Example

%NEW G71 *	Program begin, name, unit of measure
N10 BLK FORM ROTATION Z DIM_R LBL1*	Spindle axis, manner of interpretation, subprogram number
N20 M30*	End of main program
N30 G98 L1*	Subprogram start
N40 G01 X+0 Z+1*	Starting point of contour
N50 G01 X+50*	Programming in the positive direction of the principal axis
N60 G01 Z-20*	
N70 G01 X+70*	
N80 G01 Z-100*	
N90 G01 X+0*	
N100 G01 Z+1*	Contour end
N110 G98 L0 *	End of subprogram
N99999999 %NEW G71 *	Program end, name, unit of measure

Creating a new NC program



You always enter an NC program in **Programming** mode. An example of program initiation:

- 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.I

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

G17

- Enter the spindle axis, e.g. **G17**

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

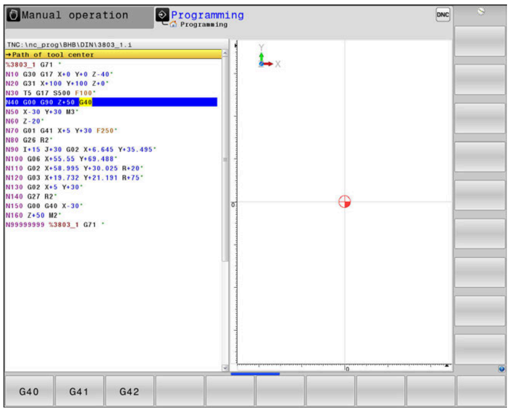
%NEW G71 *	Program begin, name, unit of measure
N10 G30 G17 X+0 Y+0 Z-40*	Spindle axis, MIN point coordinates
N20 G31 X+100 Y+100 Z+0*	MAX point coordinates
N99999999 %NEW G71 *	Program end, name, unit of measure

The control automatically generates the first and last NC blocks of the NC program.

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

Programming tool movements in DIN/ISO

To program an NC block, pressing the **SPEC FCT** key. Press the **PROGRAM FUNCTIONS** soft key, and then the **DIN/ISO** soft key. You can also use the gray path function keys to get the corresponding G code.



> If you enter ISO functions on a keyboard connected through a USB port, make sure that capitalization is active.

Example of a positioning block

G ENT

- Press the **G** key
- Enter **1** and press the **ENT** key to open the NC block

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**.

Path of tool center

G
- Enter **40** and confirm with the **ENT** key to traverse without tool radius compensation

Alternative:

G41 G42
- Move the tool to the left or to the right of the programmed contour: Press the **G41** or **G42** soft key

Feed rate F=?

- **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

```
N30 G01  G40 X+10  Y+5 F100 M3*
```

Actual position capture

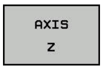
The control enables you to transfer the current tool position into the NC program,for example during

- 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.



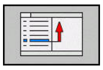
- ▶ Select the axis
- > The control writes the current position of the selected axis into the active input box.

> 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 if 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
GOTO □	Select a specific NC block **Further information:** "Using the GOTO key", Page 182

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.
- Select the target directory if required with the **SWITCH** soft key
- Enter the file name
- Confirm with the **OK** soft key or the **ENT** key, or press the **CANCEL** soft key to abort

> The file saved with **SAVE AS** can also be found in the file management 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

CANCEL CHANGE

- Press the **CANCEL CHANGE** soft key
- > The control opens a window in which you can confirm or cancel this action.
- Confirm with the **YES** soft key or cancel with the **ENT** key, or press the **NO** soft key to abort

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

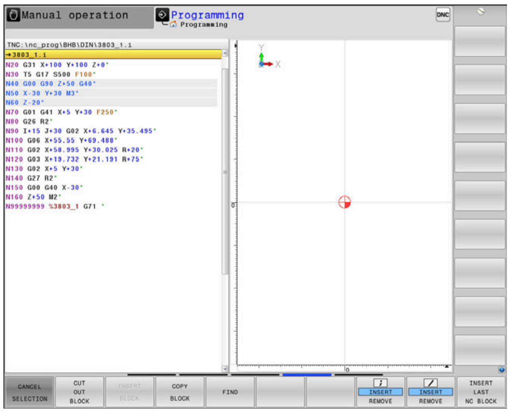
- Select a word in an NC block: Press the arrow key repeatedly until the desired word is highlighted
- Select an NC block with the arrow keys
  - Arrow down: search forwards
  - Arrow up: search backwards

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

> 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.

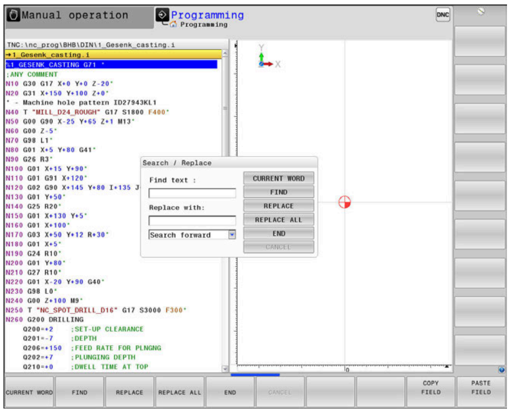
> If you want to transfer a program section to another NC program, you now need to select the desired NC program in the file manager.

- ▶ 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
- ▶ To 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 found syntax elements without a confirmation prompt. The original file is not automatically backed up by the control before the replacement process. As a consequence, NC programs may be irreversibly damaged.

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

> The **FIND** and **REPLACE** functions cannot be used in the active NC program while the program is being run. The 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. To replace all text occurrences, press the **REPLACE ALL** soft key. 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	.I

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. Table names must start with a letter.

> 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 102

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 ****.

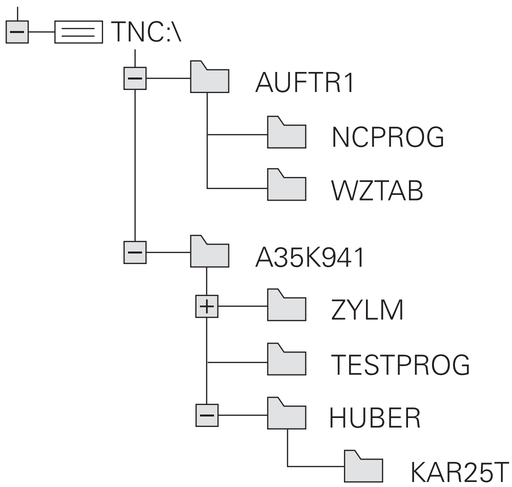
> The maximum permitted path length is 255 characters. The path length consists of the drive characters, the directory name and the file name, including the extension.

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:

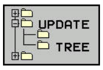
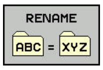
TNC:\AUFTR1\NCPROG\PROG1.I

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



Overview: Functions of the file manager

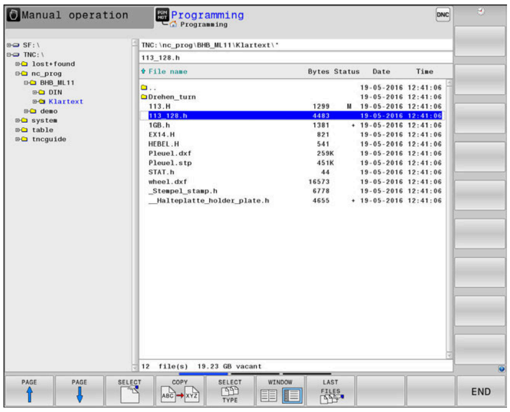
Soft key	Function	Page
COPY ABC → XYZ	Copy a single file	108
SELECT TYPE	Display a specific file type	106
NEW FILE	Create new file	108
LAST FILES	Display the last 10 files that were selected	112
DELETE	Delete a file	113
TAG	Tag a file	114
RENAME ABC = XYZ	Rename file	115
PROTECT	Protect a file against editing and erasure	116
UNPROTECT	Cancel file protection	116
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	341
NET	Manage network drives	See the User's Manual for Setup, Testing and Running NC Programs
SELECT EDITOR	Select the editor	116
SORT	Sort files by properties	115
COPY DIR	Copy a directory	112
DELETE ALL	Delete directory with all its subdirectories	

Soft key	Function	Page
	Refresh directory	
	Rename a 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

- To call the file manager, press 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

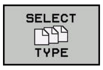
ENT

- Press the **ENT** key

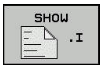
Step 2: Select a directory

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

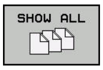
Step 3: Select a file



- Press the **SELECT TYPE** soft key



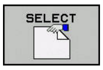
- Press the soft key for the desired file type, or



- Display all files: Press the **SHOW ALL** soft key, or

- Use wildcards, e.g. **4*.h**: Show all files of type .h starting with a 4

- Move the highlight to the desired file in the right window



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

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

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

Creating a new directory

- Move the highlight in the left window to the directory in which you want to create a subdirectory

NEW DIRECTORY

- Press the **NEW DIRECTORY** soft key
- Enter a directory name

ENT

- Press the **ENT** key

OK

- Press the **OK** soft key to confirm or

CANCEL

- 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

NEW FILE

- Press the **NEW FILE** soft key
- Enter the file name with extension

ENT

- Press the **ENT** key

Copying a single file

- Move the cursor to the file you wish to copy

COPY ABC XYZ

- 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.

OK

- 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

OK

- 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.

> 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

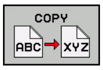
- Press the **SHOW TREE** soft key
- Move the cursor to the directory into which you wish to copy the files, and display the files in this directory with the **ENT** key

In the left window

- Press the **SHOW TREE** soft key
- Select the directory with the files to copy and press the **SHOW FILES** soft key to display them

- 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 114

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! If you use the **REPLACE FIELDS** function, all lines of the target file that are contained in the copied table will be overwritten 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 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.

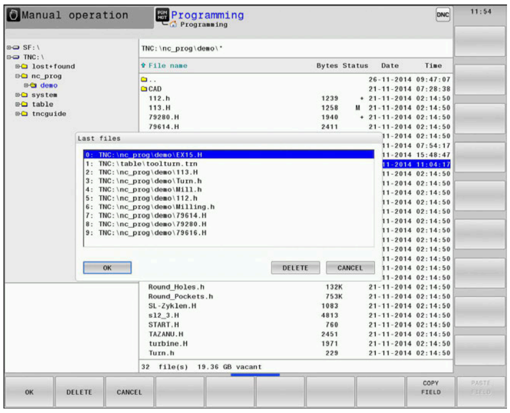
Proceed as follows:

- 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

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

Deleting a file

NOTICE

Caution: Data may be lost!

The **DELETE** function permanently deletes the file. The file is not automatically backed up by the control, e.g. to a recycle bin, before being deleted. Files are irreversibly deleted by this function.

- 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 files are not automatically backed up by the control, e.g. to a recycle bin, before being deleted. Files are irreversibly deleted by this function.

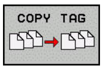
- Regularly back up important data to external drives

Proceed as follows:

- Move the cursor to the directory you want to delete

- Press the **DELETE** 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

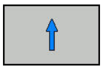
Soft key	Tagging function
	Tag a single file
	Tag all files in the directory
	Untag a single file
	Untag all files
	Copy all tagged files

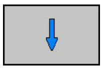
Some functions, such as copying or erasing files, can not only be used for individual files, but also for several files at once. To tag several files, proceed as follows:

- 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

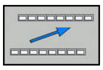




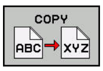
- Move the cursor to other files

- To select the next file, press the **TAG FILE** soft key. Repeat this process for all files you want to tag.

To copy tagged files:

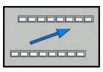


- Leave the active soft-key row



- Press the **COPY** soft key

To delete tagged files:

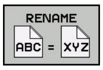


- Leave the active soft-key row

- Press the **DELETE** soft key

Renaming a file

- Move the cursor to the file you wish to rename



- To 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

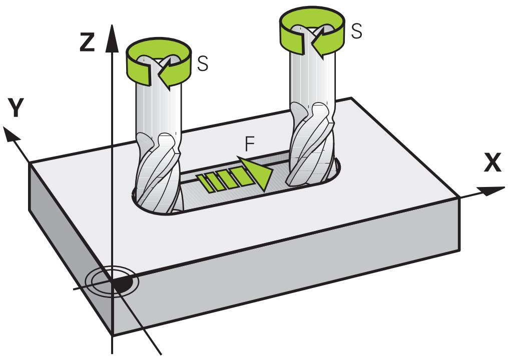
4

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 **T** block and in every positioning block.

Further information: "Programming tool movements in DIN/ISO", Page 91

You enter the feed rate **F** in mm/min in millimeter programs, and in 1/10 inch/min in inch-programs, for resolution reasons.

Rapid traverse

If you wish to program rapid traverse, enter **G00.**

> To move your machine at rapid traverse, you can also program the corresponding numerical value, e.g. **G01 F30000**. Unlike **G00**, 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. **G00** is only effective in the NC block in which it is programmed. After the NC block with **G00** 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 **T** 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 **T** block by entering only the new spindle speed.

Proceed as follows:

S

- Press the **S** key on the alphabetic keyboard
- Enter the new spindle speed

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

Changing during program run

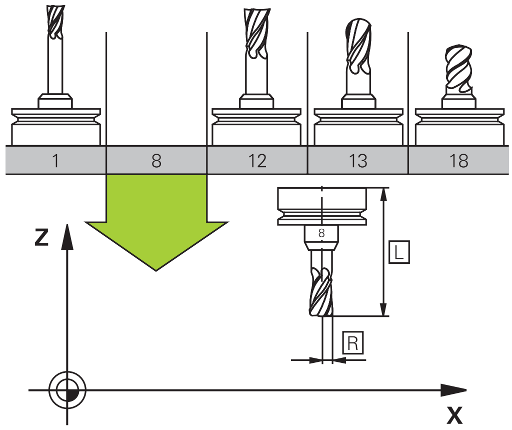
You can adjust the spindle speed during program run with the spindle speed potentiometer S.

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 **G99** 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

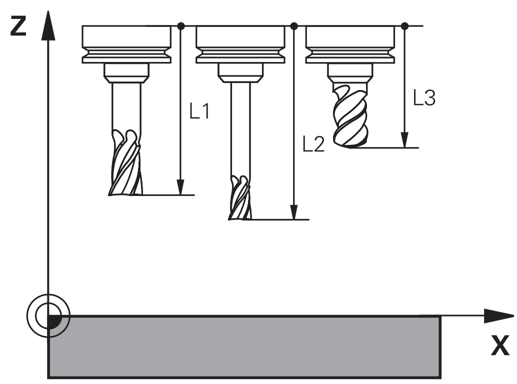
Each tool is identified by a number between 0 and 32767. If you are working with tool tables, you can also enter a tool name for each tool. Tool names can have up to 32 characters.

> **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

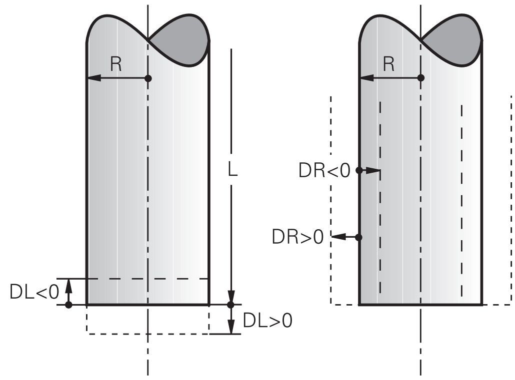
You should always enter the tool length L as an absolute value based on the tool reference point. The entire tool length is essential for the control in order to perform numerous functions involving multi-axis machining.



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 describes a tool oversize (**DL**, **DR**>0). If you are programming the machining data with an allowance, enter the oversize value in the **T**.

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 **T** 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 **T** block do not change the represented size of the **tool** during the simulation. However, the programmed delta values move the **tool** by the defined value in the simulation.

> Delta values from the **T** 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 **G99** function.

The number, length, and radius of a specific tool are defined in the **G99** block of the NC program:

Proceed as follows for the definition:

TOOL DEF

- press the **TOOL DEF** key.
- **Tool length**: Compensation value for the tool length
- **Tool radius**: Compensation value for the tool radius

Example

```
N40 G99 T5 L+10 R+5*
```

Calling the tool data

Before you can call the tool, you have to define it in a **G99** block or in the tool table.

A **T** 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 places the tool name in quotation marks. You have to assign a tool name to a string parameter first. Names always 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). The feed rate is effective until you program a new feed rate in a positioning block or in a **T** 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:
>
> - **T** block without tool name, tool number, and tool axis
> - **T** block without tool name and tool number, and with the same tool axis as in the previous **T** block
>
> In the following cases the control runs the tool-change macro and inserts a replacement tool if necessary:
>
> - **T** block with tool number
> - **T** block with tool name
> - **T** 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

```
N20 T  5.2 G17  S2500 DL+0.2  DR-1*
```

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

Preselection of tools

> Refer to your machine manual.
> The preselection of tools with **G51** can vary depending on the individual machine tool.

If you are working with tool tables, use a **G51** block to preselect the next tool. Simply enter the tool number, or a Q 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 **T**, 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 (**G41**/**G42**) is active
- Directly after an approach function **APPR**
- Directly before a departure function **DEP**
- Directly before and after **G24** and **G25**
- During execution of macros
- During execution of a tool change
- Directly after a **T** block or **G99**
- 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 **T** 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.**

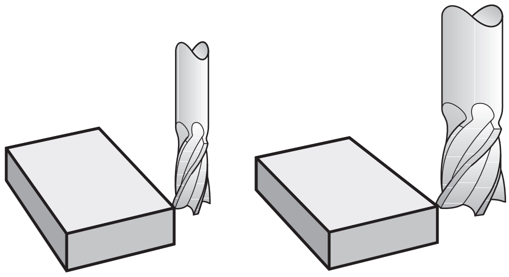
4.3 Tool compensation

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. **T 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 **T 0**. Danger of collision during subsequent tool positioning movements!

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

For tool length compensation, the control takes the delta values from both the **T** block and the tool table into account:

Compensation value = **L** + $\mathbf{DL}_{\text{CALL T block}}$ + $\mathbf{DL}_{\text{TAB}}$ with

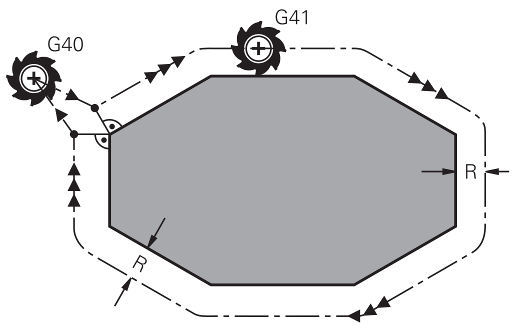
L:	Tool length **L** from **G99** block or tool table
$\mathbf{DL}_{\text{CALL T block}}$:	Oversize for length **DL** in the **T** block
$\mathbf{DL}_{\text{TAB}}$:	Oversize for length **DL** in the tool table

Tool radius compensation

The block for programming a tool movement contains:

- **G41** or **G42** for radius compensation
- **G40**, if there is no radius compensation

The radius compensation is effective as soon as a tool is called and traversed with a straight-line block in the working plane with **G41** or **G42**.



> The control automatically cancels radius compensation in the following cases:
> - Straight-line block with **G40**
> - **DEP** function for departing from the contour
> - Select a new NC program via **PGM MGT**

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

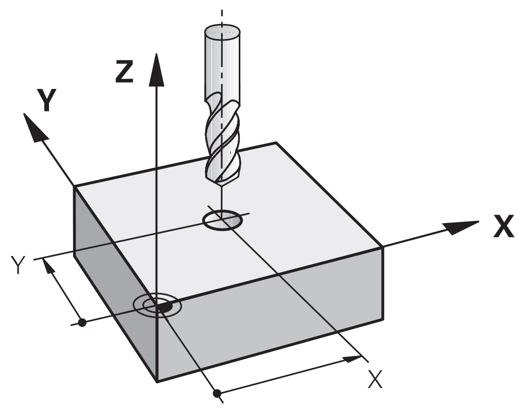
Compensation value = **R** + $\mathbf{DR}_{\text{CALLT block}}$ + $\mathbf{DR}_{\text{TAB}}$ with

R:	Tool radius **R** from **G99** block or tool table
$\mathbf{DR}_{\text{CALLT block}}$:	Oversize for radius **DR** in the **T** block
$\mathbf{DR}_{\text{TAB}}$:	Oversize for radius **DR** in the tool table

Contouring without radius compensation: G40

The tool center moves in the working plane along the programmed path, or to the programmed coordinates.

Applications: Drilling and boring, pre-positioning

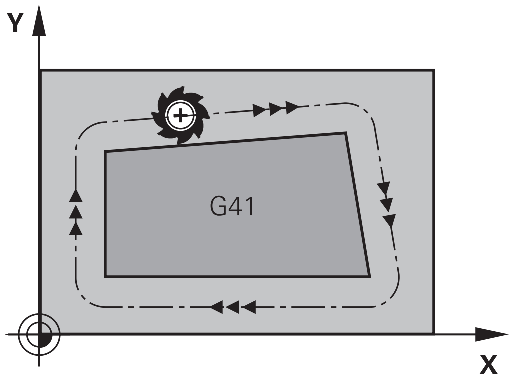


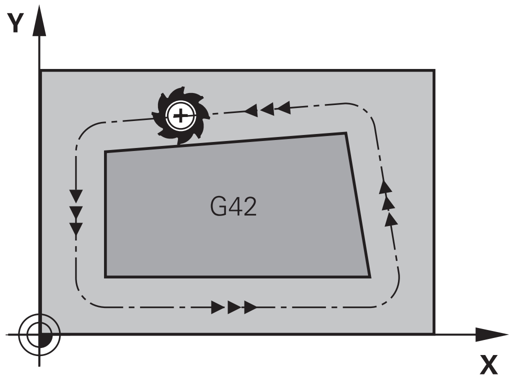
Contouring with radius compensation: G42 and G41

G42: The tool moves to the right of the programmed contour

G41: 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 **G42** and **G41** you must program at least one traversing block in the working plane without radius compensation (that is, with **G40**).
>
> 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/RLG42/G41** or canceled with **G40** the control always positions the tool perpendicular to the programmed starting or end position. Position the tool at a sufficient distance from the first or last contour point to prevent the possibility of damaging the contour.

Entering radius compensation

Radius compensation is entered in a **G01** block. Enter the coordinates of the target point and confirm your entry with the **ENT** key.

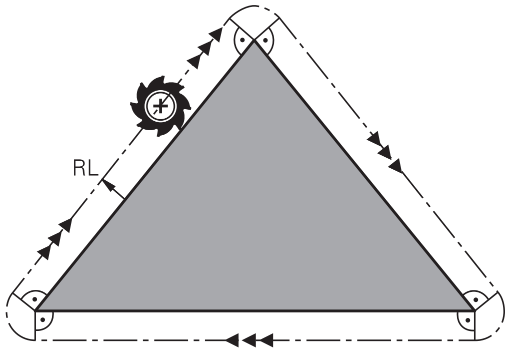
G41
- Select tool movement to the left of the programmed contour: Press the **G41** soft key, or

G42
- Select tool movement to the right of the contour: Press the **G42** soft key, or

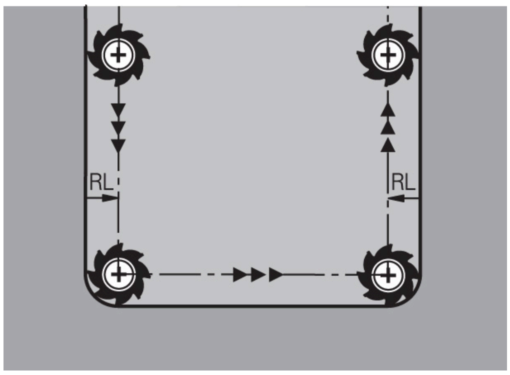
G40
- Select tool movement without radius compensation or cancel radius compensation: Select function **G40**

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

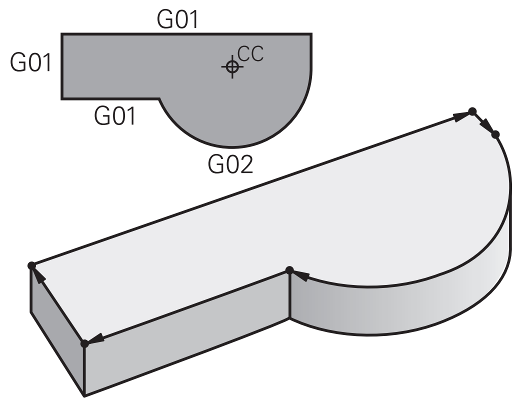
5

Programming Contours

5.1 Tool movements

Path functions

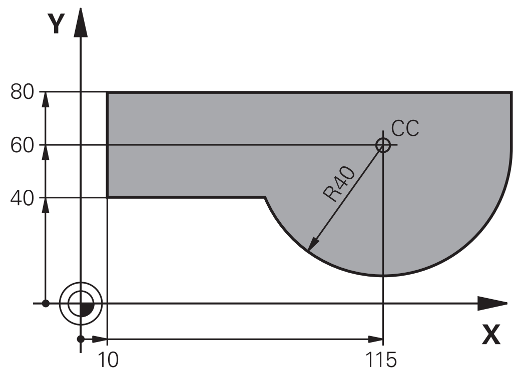
A workpiece contour is usually composed of several contour elements such as straight lines and circular arcs. With the path functions, you can program the tool movements for **straight lines** and **circular arcs**.



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 235

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 255

5.2 Fundamentals of path functions

Programming tool movements for workpiece machining

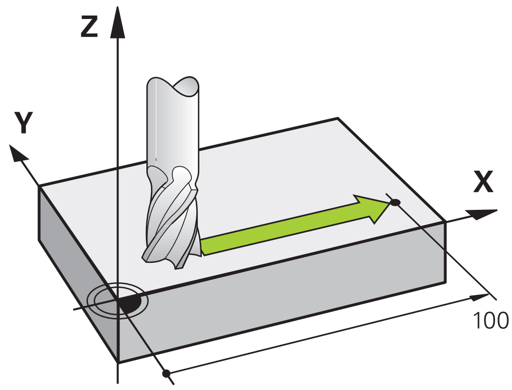
You create an NC program by programming the path functions for the individual contour elements in sequence You do this by entering the coordinates of the end points of the contour elements given in the production drawing. The control calculates the actual path of the tool from these coordinates, and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

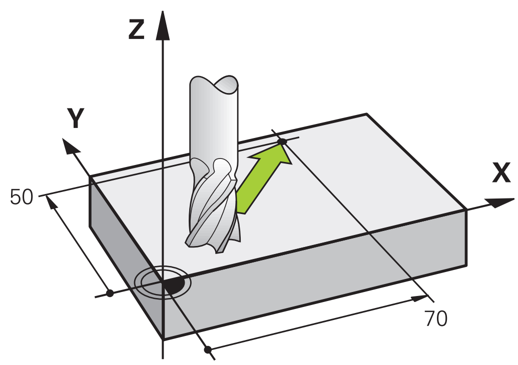
```
N50 G00 X+100*
```

N50	Block number
G00	Path function **straight line at rapid traverse**
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.



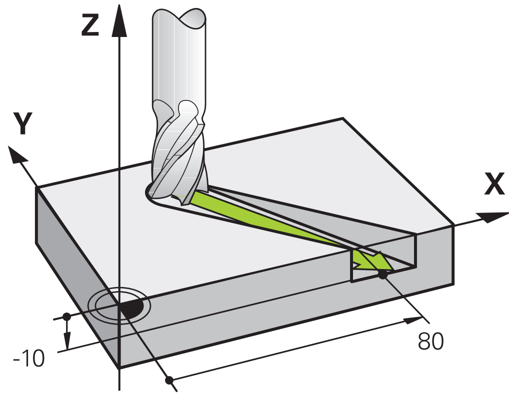
Example

```
N50 G00 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

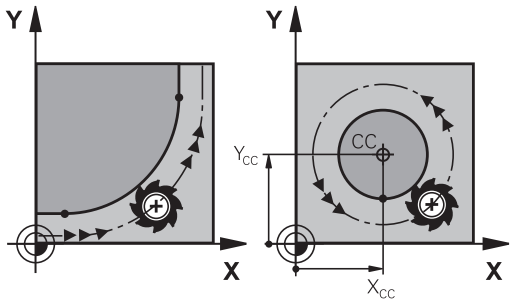
```
N50 G01 X+80  Y+0 Z-10*
```

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

Example

```
N50 G01 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 with **I** and **J**.

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 **T**:

Spindle axis	Main plane
(G17)	**XY**, also UV, XV, UY
(G18)	**ZX**, also WU, ZU, WX
(G19)	**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 353
>
> **Further information:** "Principle and overview of functions", Page 256

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: **G02/G12**

Counterclockwise direction of rotation: **G03/G13**

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 148

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

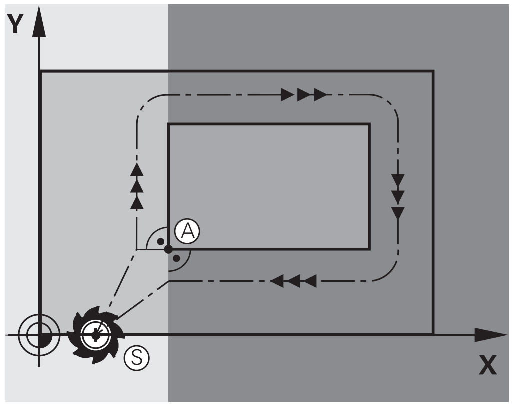
5.3 Approaching and departing a contour

Starting point and end point

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

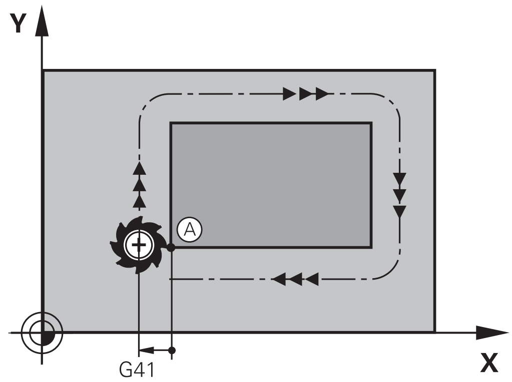
- Programmed without radius compensation
- Approachable without danger of collision
- Close to the first contour point

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.



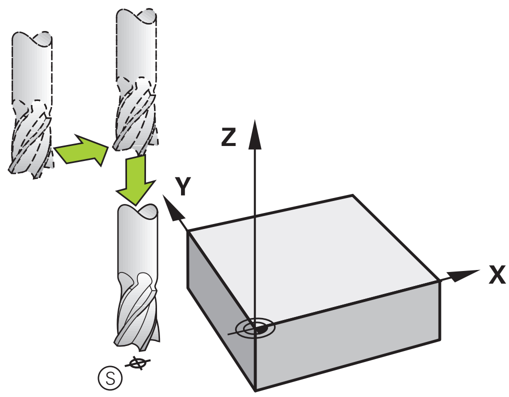
First contour point

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



Approaching the starting point in the spindle axis

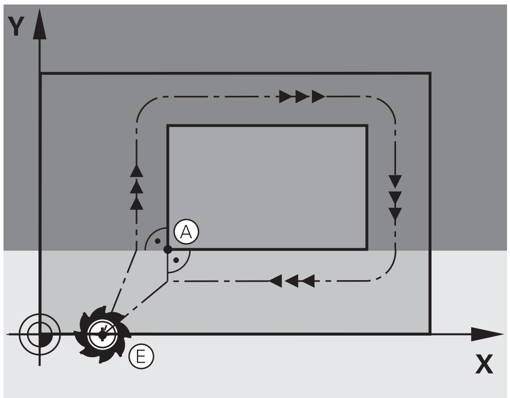
When the starting point is approached, the tool must be moved to the working depth in the spindle axis. If danger of collision exists, approach the starting point in the spindle axis separately.



Example

```
N40 G00  Z-10*
N30 G01  X+20  Y+30  G41 F350*
```

End point

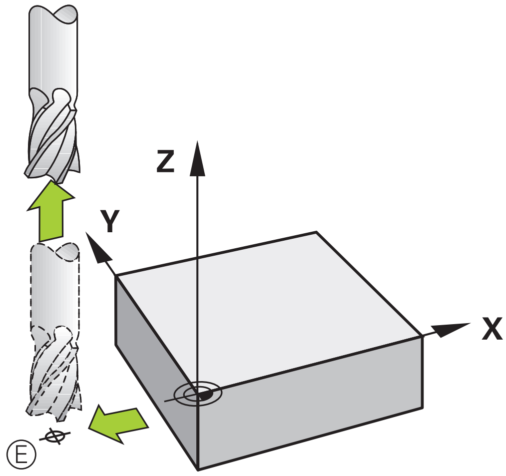


The end point should be selected so that it is:

- 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.

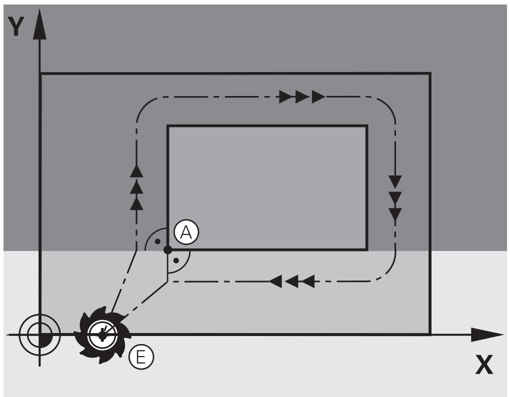
Departing the end point in the spindle axis:
Program the departure from the end point in the spindle axis separately.



Example

N50 G01 G40 X+60 Y+70 F700*
N60 G00 Z+250*

Common starting and end points



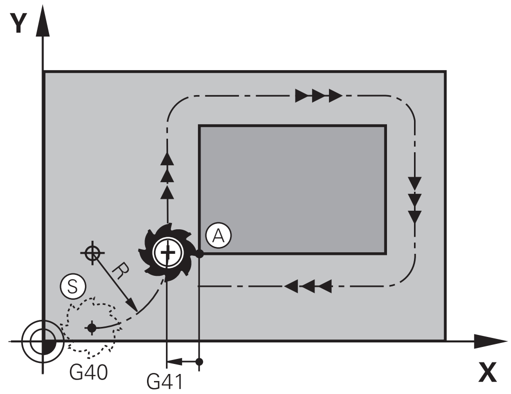
Do not program any radius compensation if the starting point and end point are the same.

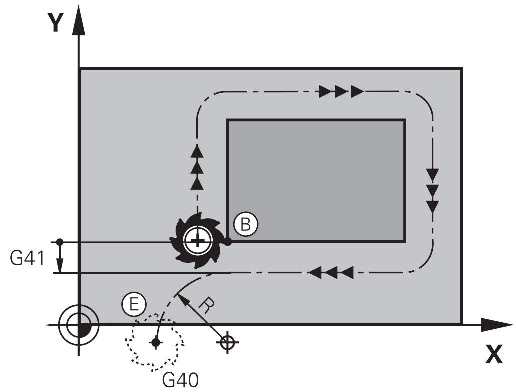
In order to make sure the contour will not be damaged, the optimal starting point should lie between the extended tool paths for machining the first and last contour elements.

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.

Tangential approach and departure

With **G26** (figure at center right), you can program a tangential approach to the workpiece, and with **G27** (figure at lower right) a tangential departure. In this way you can avoid dwell marks.





Starting point and end point

The starting point and the end point lie outside the workpiece, close to the first and last contour points. They are to be programmed without radius compensation.

Approach

- **G26** is entered after the NC block in which the first contour element is programmed: This will be the first NC block with radius compensation **G41/G42**

Departure

- **G27** after the NC block in which the last contour element is programmed: This will be the last NC block with radius compensation **G41/G42**

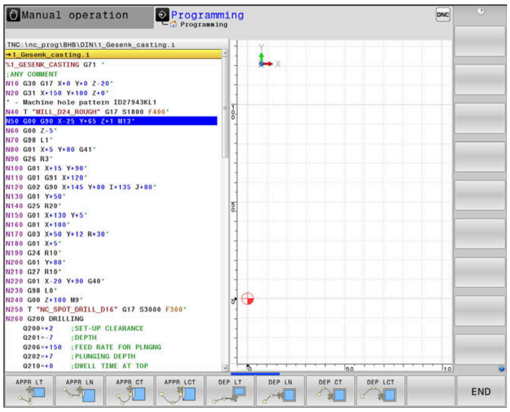
> The radius for **G26** and **G27** must be selected so that the control can execute the circular path between the starting point and the first contour point, as well as the last contour point and the end point.

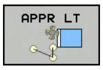
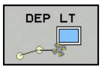
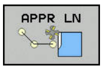
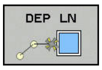
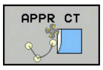
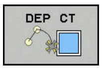
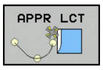
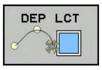
Example

N50 G00 G40 G90 X-30 Y+50*	Starting point
N60 G01 G41 X+0 Y+50 F350*	First contour point
N70 G26 R5*	Tangential approach with radius R = 5 mm
. . .	
Program contour blocks	
. . .	Last contour point
N210 G27 R5*	Tangential departure with radius R = 5 mm
N220 G00 G40 X-30 Y+50*	End point

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:

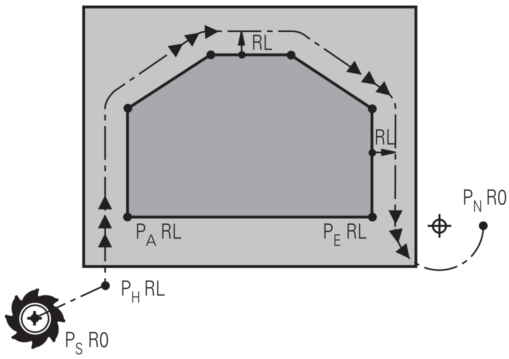


Approach	Departure	Function
		Straight line with tangential connection
		Straight line perpendicular to a contour point
		Circular arc with tangential connection
		Circular arc with tangential connection to the contour. Approach and departure to an auxiliary point outside the contour on a tangentially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



R0=G40; RL=G41; RR=G42

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 **G00** 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 **G00** 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 (G40).
- 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

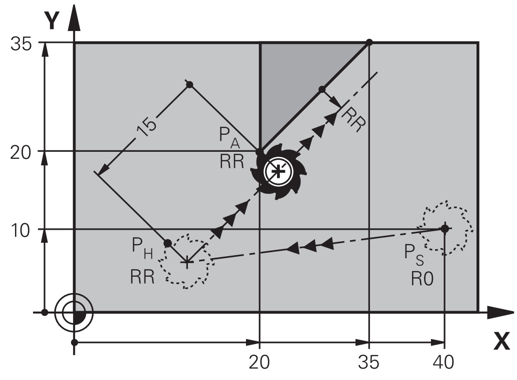
The tool radius compensation is programmed together with the first contour point P_A in the APPR block. The DEP blocks automatically discard the tool radius compensation.

If you program **APPR LN** or **APPR CT** with **G40**, the control stops the machining/simulation with an error message.

This method of function differs from the iTNC 530 control!

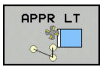
Approaching on a straight line with tangential connection: APPR LT

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**.



R0=G40; RL=G41; RR=G42

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



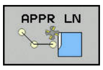
  - Coordinates of the first contour point P_A
  - **LEN**: Distance from the auxiliary point P_H to the first contour point P_A
  - Radius compensation **G41/G42** for machining

Example

N70 G00 X+40 Y+10 G40 M3*	Approach P_S without radius compensation
N80 APPR LT X+20 Y+20 Z-10 LEN15 G42 F100*	P_A with radius comp. G42, distance P_H to P_A: LEN=15
N90 G01 X+35 Y+35*	End point of the first contour element
N100 G01 ...*	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 **G41/G42** for machining

Example

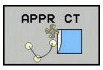
N70 G00 X+40 Y+10 G40 M3*	Approach PS without radius compensation
N80 APPR LN X+10 Y+20 Z-10 LEN15 G24 F100*	PA with radius comp. G42
N90 G01 X+20 Y+35*	End point of the first contour element
N100 G01 ...*	Next contour element

Approaching on a circular path with tangential connection: APPR CT

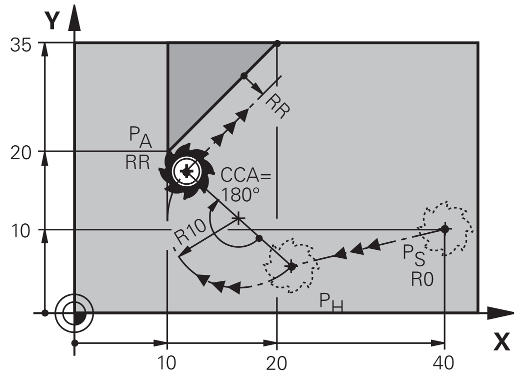
The tool moves on a straight line from the starting point P_S to an auxiliary point P_H. It then moves from PH to the first contour point PA following a circular arc that is tangential to the first contour element.

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 **G41/G42** for machining

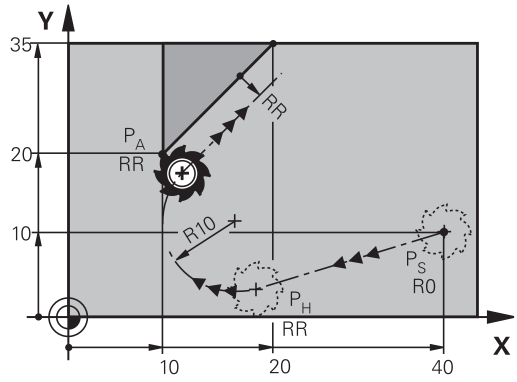


R0=G40; RL=G41; RR=G42

Example

N70 G00 X+40 Y+10 G40 M3*	Approach PS without radius compensation
N80 APPR CT X+10 Y+20 Z-10 CCA180 R+10 G42 F100*	PA with radius comp. G42, radius R=10
N90 G01 X+20 Y+35*	End point of the first contour element
N100 G01 ...*	Next contour element

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



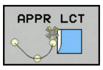
R0=G40; RL=G41; RR=G42

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 **G41/G42** for machining

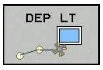
Example

N70 G00 X+40 Y+10 G40 M3*	Approach PS without radius compensation
N80 APPR LCT X+10 Y+20 Z-10 R10 G42 F100*	PA with radius comp. G42, radius R=10
N90 G01 X+20 Y+35*	End point of the first contour element
N100 G01 ...*	Next contour element

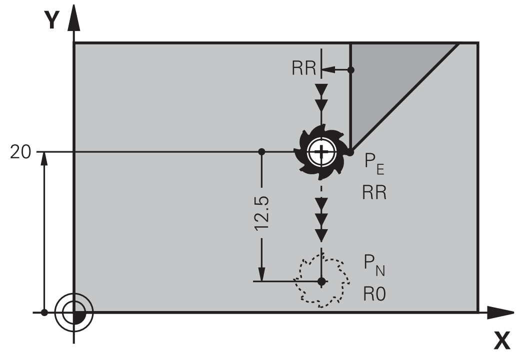
Departing in a straight line with tangential connection: DEP LT

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.



R0=G40; RL=G41; RR=G42

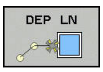
Example

N20 G01 Y+20 G42 F100*	Last contour element: PE with radius compensation
N30 DEP LT LEN12.5 F100*	Depart contour by LEN=12.5 mm
N40 G00 Z+100 M2*	Retract in Z, return jump, end program

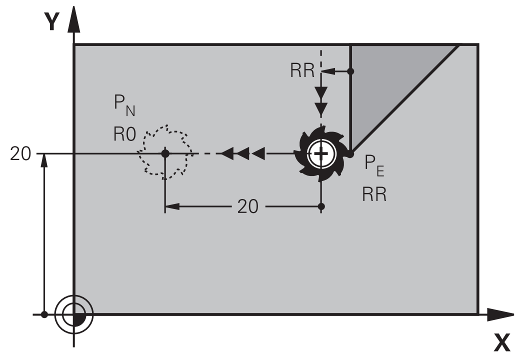
Departing in a straight line perpendicular to the last contour point: DEP LN

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**



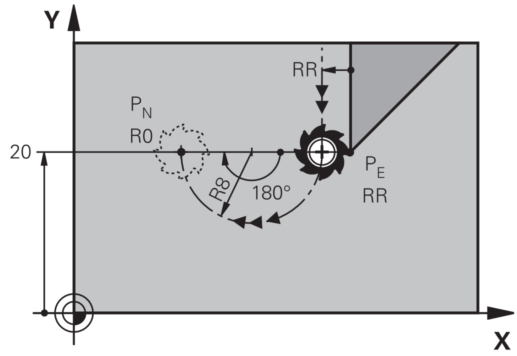
R0=G40; RL=G41; RR=G42

Example

N20 G01 Y+20 G42 F100*	Last contour element: PE with radius compensation
N30 DEP LN LEN+20 F100*	Depart perpendicular to contour by LEN=20 mm
N40 G00 Z+100 M2*	Retract in Z, return jump, end program

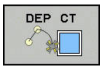
Departing on a circular path with tangential connection: DEP CT

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.



R0=G40; RL=G41; RR=G42

- 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



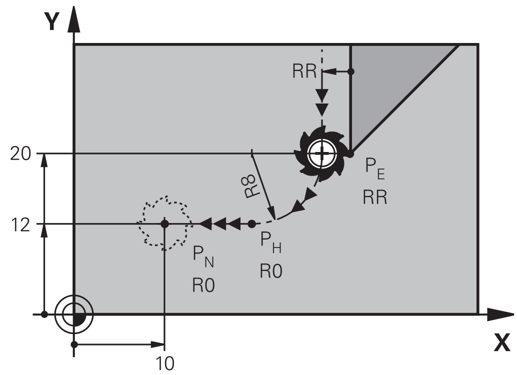
- Center angle **CCA** of the arc
- Radius R of the circular arc
  - If the tool should depart the workpiece in the direction opposite to the radius compensation: Enter R as a positive value.
  - If the tool should depart the workpiece in the direction **opposite** to the radius compensation: Enter R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

Example

N20 G01 Y+20 G42 F100*	Last contour element: PE with radius compensation
N30 DEP CT CCA 180 R+8 F100*	Center angle=180°, arc radius=8 mm
N40 G00 Z+100 M2*	Retract in Z, return jump, end program

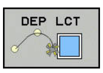
Departing on a circular arc tangentially connecting the contour and a straight line: DEP LCT

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.



R0=G40; RL=G41; RR=G42

- 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

N20 G01 Y+20 G42 F100*	Last contour element: PE with radius compensation
N30 DEP LCT X+10 Y+12 R+8 F100*	Coordinates PN, arc radius=8 mm
N40 G00 Z+100 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** **G00** and **G01**	Straight line	Coordinates of the end point	149
CHF	Chamfer: **CHF** **G24**	Chamfer between two straight lines	Chamfer side length	150
	Circle center **CC** **I** and **J**	None	Coordinates of the circle center or pole	152
C	Circular arc **C** **G02** and **G03**	Circular arc around a circle center CC to an arc end point	Coordinates of the arc end point, direction of rotation	153
CR	Circular arc **CR** **G05**	Circular arc with a certain radius	Coordinates of the arc end point, arc radius, direction of rotation	154
CT	Circular arc **CT** **G06**	Circular arc with tangential connection to the preceding and subsequent contour elements	Coordinates of the arc end point	156
RND	Corner rounding **RND** **G25**	Circular arc with tangential connection to the preceding and subsequent contour elements	Rounding radius R	151
FK	**FK** free contour programming	Straight line or circular path with any connection to the preceding contour element	Input depends on the function	170

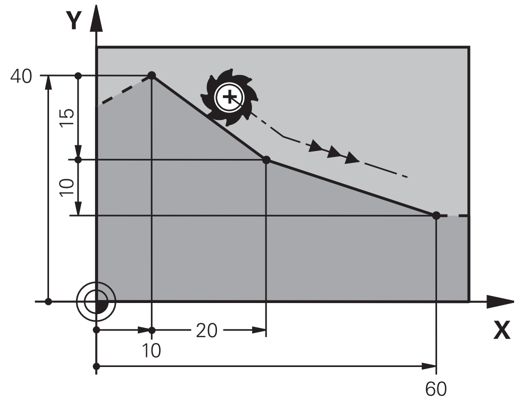
Programming path functions

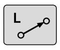
You can program path functions conveniently by using the gray path function keys. In further dialogs, you are prompted by the control to make the required entries.

> If you enter ISO functions on a keyboard connected through a USB port, make sure that capitalization is active.
> At the start of the block the control automatically writes in capitals.

Straight line in rapid traverse G00 or straight line with feed rate F G01

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 with feed rate
- **Coordinates** of the end point of the straight line, if necessary
- **Radius compensation G40/G41/G42**
- **Feed rate F**
- **Miscellaneous function M**

Movement at rapid traverse

A straight line block for a rapid traverse motion (**G00** block) can also be initiated with the **L** key:

- Press the **L** key to open a program block for a linear movement
- Press the left arrow key to switch to the input range for G codes
- Press the **G00** soft key if you want to enter a rapid traverse motion

Example

N70 G01 G41 X+10 Y+40 F200 M3*
N80 G91 X+20 Y-15*
N90 G90 X+60 G91 Y-10*

Actual position capture

You can also generate a straight-line block (**G01** block) by using the **actual position capture** key:

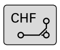
- In the **Manual Operation** mode, move the tool to the position you want to capture
- Switch the screen display to programming
- Select the NC block after which you want to insert the straight line block

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

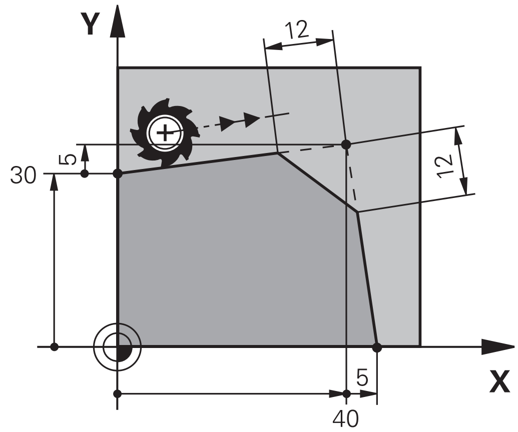
Inserting a chamfer between two straight lines

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

- The line blocks before and after the **G24** block must be in the same working plane as the chamfer.
- The radius compensation before and after the **G24** block must be the same
- The chamfer must be machinable with the current tool



- **Chamfer side length**: Length of the chamfer, and if necessary:
- **Feed rate F** (effective only in **G24** block)



Example

```
N70 G01  G41 X+0  Y+30 F300  M3*
N80 X+40  G91 Y+5*
N90 G24  R12 F250*
N100 G91  X+5 G90  Y+0*
```

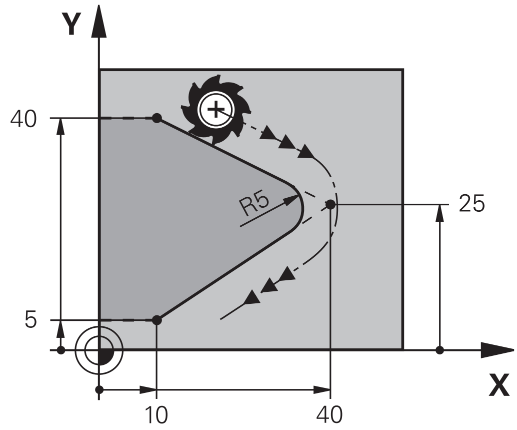
> You cannot start a contour with a **G24** block.
> A chamfer is possible only in the working plane.
> The corner point is cut off by the chamfer and is not part of the contour.
> A feed rate programmed in the **G24** block is effective only in that CHF block. After the **G24** block, the previous feed rate becomes effective again.

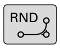
Rounded corners G25

The **G25** 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 **G25** block)

Example

N50 G01 X+10 Y+40 G41 F300 M3*
N60 G01 X+40 Y+25*
N70 G25 R5 F100*
N80 G01 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.

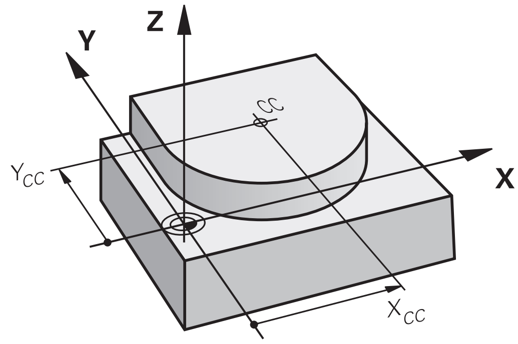
A feed rate programmed in the **G25** block is effective only in that **G25** block. After the **G25** block, the previous feed rate becomes effective again.

You can also use an **G25** block for a tangential contour approach.

Circle centerI, J

You can define a circle center for circles that you have programmed with the **G02**, **G03**, or **G05** function. 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



SPEC FCT

- ▶ To program the circle center, press the **SPEC FCT** key
- ▶ Press the **PROGRAM FUNCTIONS** soft key
- ▶ Press the **DIN/ISO** soft key
- ▶ Press the **I** or **J** soft key
- ▶ Enter coordinates for the circle center or, if you want to use the last programmed position, enter **G29**

Example

```
N50 I+25  J+25*
```

or

```
N10 G00 G40  X+25 Y+25*
N20 G29*
```

The program lines 10 and 20 do not refer to the illustration.

Validity

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

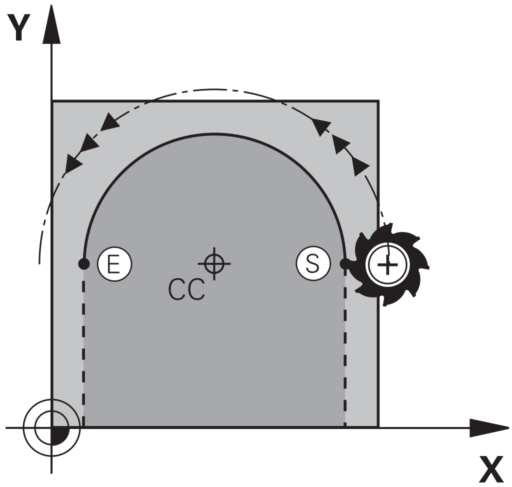
Entering the circle center incrementally

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 **I** and **J** 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 around circle center

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



Direction of rotation

- In clockwise direction: **G02**
- In counterclockwise direction: **G03**
- Without programmed direction: **G05**. The control traverses the circular arc with the last programmed direction of rotation.

▶ Move the tool to the starting point of the circle

J

I

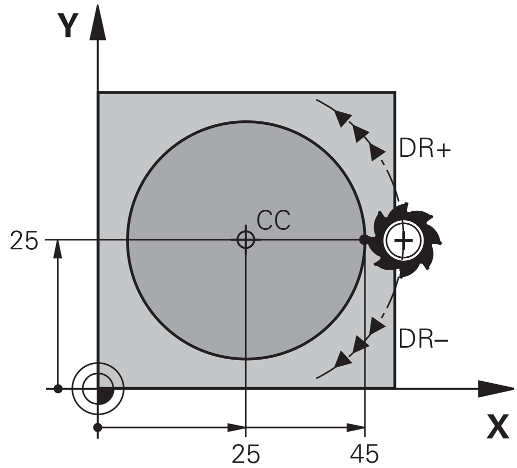
▶ Enter the **coordinates** of the circle center

C

▶ Enter the **coordinates** of the arc end point, if necessary:

▶ **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. **G2 Z... X...** (with tool axis Z).

Example

N50 I+25 J+25*
N60 G01 G42 X+45 Y+25 F200 M3*
N70 G03 X+45 Y+25*

Full circle

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

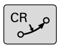
> The starting and end points of the arc must lie on the circle.
>
> The maximum value for input tolerance is 0.016 mm. Set the input tolerance in the machine parameter **circleDeviation** (no. 200901).
>
> Smallest possible circle that the control can traverse: 0.016 mm.

Circular arc G02/G03/G05 with fixed radius

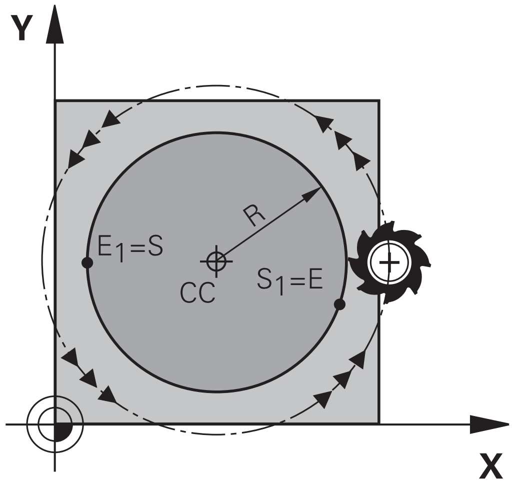
The tool moves on a circular path with the radius R.

Direction of rotation

- In clockwise direction: **G02**
- In counterclockwise direction: **G03**
- Without programmed direction: **G05**. The control traverses the circular arc with the last programmed direction of rotation.



- **Coordinates** of the arc end point
- **Radius R** Caution: The algebraic sign determines the size of the arc!
- **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°

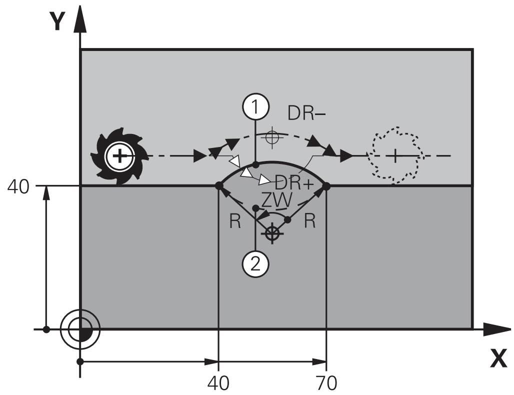
Enter the radius with a positive sign, i.e. R>0

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 **G02** (with radius compensation **G41**)

Concave: Direction of rotation **G03** (with radius compensation **G41**)



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

```
N100 G01 G41  X+40 Y+40  F200 M3*
N110 G02  X+70 Y+40  R+20*  (arc 1)
```

or

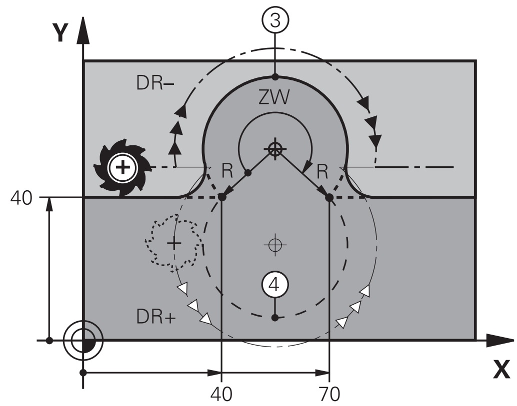
```
N110 G03  X+70 Y+40  R+20*  (arc 2)
```

or

```
N110 G02  X+70 Y+40  R-20*  (arc 3)
```

or

```
N110 G03  X+70 Y+40  R-20*  (arc 4)
```

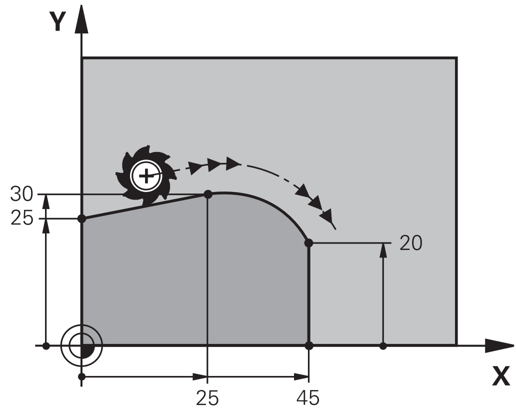


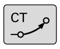
Circular arc G06 with tangential transition

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 **G06** block. This requires at least two positioning blocks.





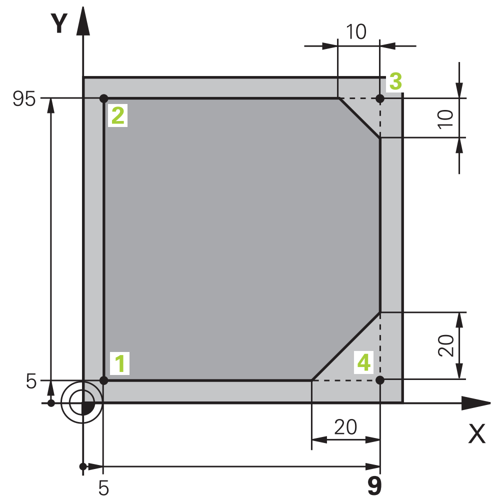
- **Coordinates** of the arc end point, and if necessary:
- **Feed F**
- **Miscellaneous function M**

Example

N70 G01 G41 X+0 Y+25 F300 M3*
N80 X+25 Y+30*
N90 G06 X+45 Y+20*
N100 G01 Y+0*

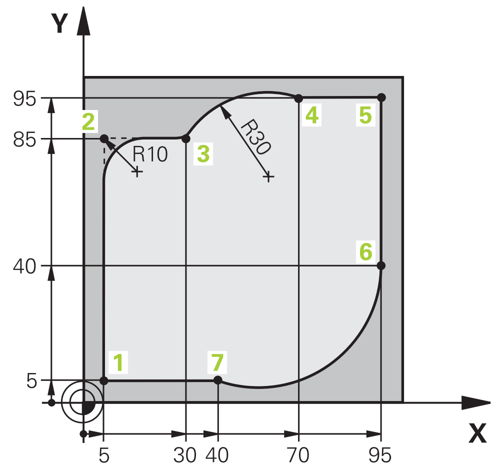
A tangential arc is a two-dimensional operation: the coordinates in the **G06** block and in the contour element preceding it must be in the same plane of the arc!

Example: Linear movements and chamfers with Cartesian coordinates



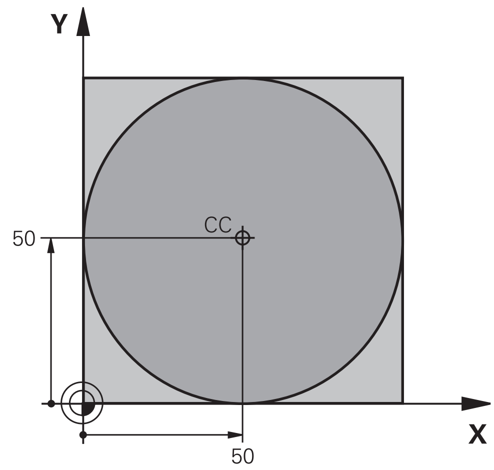
%LINEAR G71 *	
N10 G30 G17 X+0 Y+0 Z-20*	Define the workpiece blank for graphic workpiece simulation
N20 G31 G90 X+100 Y+100 Z+0*	
N30 T1 G17 S4000*	Call the tool in the spindle axis and with the spindle speed S
N40 G00 G40 G90 Z+250*	Retract the tool in the spindle axis at rapid traverse
N50 X-10 Y-10*	Pre-position the tool
N60 G01 Z-5 F1000 M3*	Move to working depth at feed rate F = 1000 mm/min
N70 G01 G41 X+5 Y+5 F300*	Approach the contour at point 1, activate radius compensation G41
N80 G26 R5 F150*	Tangential approach
N90 Y+95*	Move to point 2
N100 X+95*	Point 3: first straight line for corner 3
N110 G24 R10*	Program a chamfer with length 10 mm
N120 Y+5*	Point 4: 2nd straight line for corner 3, 1st straight line for corner 4
N130 G24 R20*	Program a chamfer with length 20 mm
N140 X+5*	Move to last contour point 1, second straight line for corner 4
N150 G27 R5 F500*	Tangential exit
N160 G40 X-20 Y-20 F1000*	Retract the tool in the working plane, cancel radius compensation
N170 G00 Z+250 M2*	Retract the tool, end program
N99999999 %LINEAR G71 *	

Example: Circular movements with Cartesian coordinates



%CIRCULAR G71 *	
N10 G30 G17 X+0 Y+0 Z-20*	Define the workpiece blank for graphic workpiece simulation
N20 G31 G90 X+100 Y+100 Z+0*	
N30 T1 G17 S4000*	Call the tool in the spindle axis and with spindle speed
N40 G00 G40 G90 Z+250*	Retract the tool in the spindle axis at rapid traverse
N50 X-10 Y-10*	Pre-position the tool
N60 G01 Z-5 F1000 M3*	Move to working depth at feed rate F = 1000 mm/min
N70 G01 G41 X+5 Y+5 F300*	Approach the contour at point 1, activate radius compensation G41
N80 G26 R5 F150*	Tangential approach
N90 Y+85*	Point 2: First straight line for corner 2
N100 G25 R10*	Insert radius with R = 10 mm, feed rate: 150 mm/min
N110 X+30*	Move to point 3: Starting point of the arc
N120 G02 X+70 Y+95 R+30*	Move to point 4: End point of the arc with G02, radius 30 mm
N130 G01 X+95*	Move to point 5
N140 Y+40*	Move to point 6
N150 G06 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
N160 G01 X+5*	Move to last contour point 1
N170 G27 R5 F500*	Depart the contour on a circular arc with tangential connection
N180 G40 X-20 Y-20 F1000*	Retract the tool in the working plane, cancel radius compensation
N190 G00 Z+250 M2*	Retract tool in the tool axis, end of program
N99999999 %CIRCULAR G71 *	

Example: Full circle with Cartesian coordinates



%C-CC G71 *	
N10 G30 G17 X+0 Y+0 Z-20*	Workpiece blank definition
N20 G31 G90 X+100 Y+100 Z+0*	
N30 T1 G17 S3150*	Tool call
N40 G00 G40 G90 Z+250*	Retract the tool
N50 I+50 J+50*	Define the circle center
N60 X-40 Y+50*	Pre-position the tool
N70 G01 Z-5 F1000 M3*	Move to working depth
N80 G41 X+0 Y+50 F300*	Approach starting point, radius compensation G41
N90 G26 R5 F150*	Tangential approach
N100 G02 X+0*	Move to the circle end point (= circle starting point)
N110 G27 R5 F500*	Tangential exit
N120 G01 G40 X-40 Y-50 F1000*	Retract the tool in the working plane, cancel radius compensation
N130 G00 Z+250 M2*	Retract tool in the tool axis, end of program
N99999999 %C-CC G71 *	

5.5 Path contours – Polar coordinates

Overview

With polar coordinates you can define a position in terms of its angle **H** and its distance **R** relative to a previously defined pole **I, J**.

Polar coordinates are useful with:

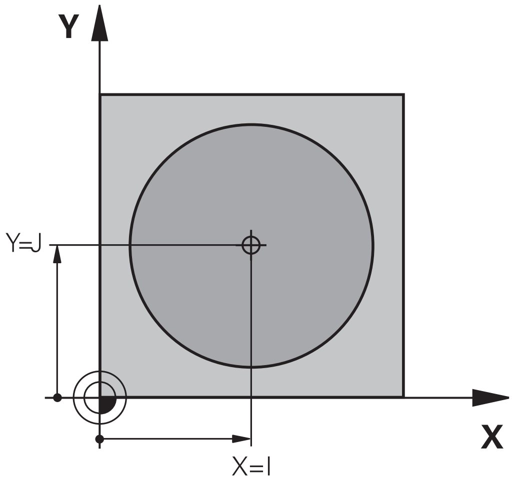
- Positions on circular arcs
- Workpiece drawing dimensions in degrees, e.g. bolt hole circles

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	161
C + P	Circular path around circle center/pole to arc end point	Polar angle of the arc end point,	162
CR + P	Circular path corresponding to active direction of rotation	Polar angle of the circle end point	162
CT + P	Circular arc with tangential connection to the preceding contour element	Polar radius, polar angle of the arc end point	162
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	163

Datum for polar coordinates: pole I, J

You can set the pole (I, J) 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.



SPEC FCT

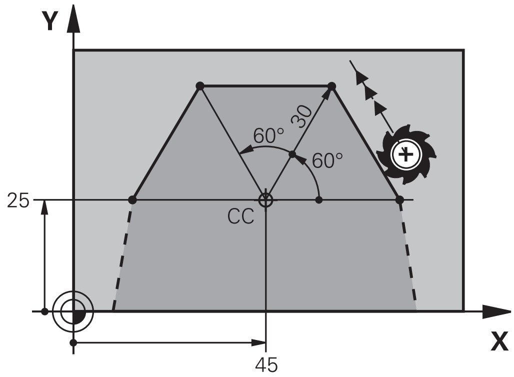
- To program a pole, press the **SPEC FCT** key.
- Press the **PROGRAM FUNCTIONS** soft key
- Press the **DIN/ISO** soft key
- Press the **I** or **J** soft key
- **Coordinates**: Enter Cartesian coordinates for the pole or, if you want to use the last programmed position, enter **G29**. Before programming 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

```
N120 I+45  J+45*
```

Straight line in rapid traverse G10 or straight line with feed rate F G11

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 R**: Enter the distance from the pole CC to the straight-line end point
- **Polar-coordinates angle H**: Angular position of the straight-line end point between –360° and +360°

The sign of **H** depends on the angle reference axis:

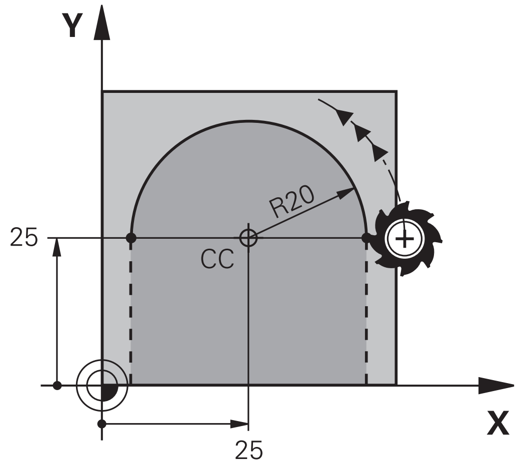
- If the angle from the angle reference axis to **R** is counterclockwise: **H**>0
- If the angle from the angle reference axis to **R** is clockwise: **H**<0

Example

```
N120 I+45  J+45*
N130 G11 G42  R+30 H+0  F300 M3*
N140 H+60*
N150 G91  H+60*
N160 G90  H+180*
```

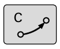
Circular path G12/G13/G15 around pole I, J

The polar coordinate radius **R** is also the radius of the arc. **R** is defined by the distance from the starting point to the pole **I, J**. The last programmed tool position will be the starting point of the arc.



Direction of rotation

- In clockwise direction: **G12**
- In counterclockwise direction: **G13**
- Without programmed direction: **G15**. The control traverses the circular arc with the last programmed direction of rotation.



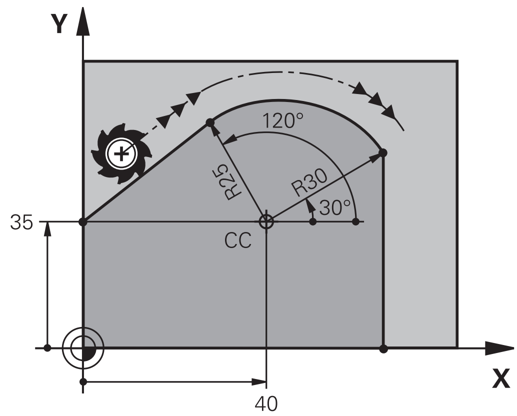
- **Polar-coordinates angle H**: Angular position of the arc end point between –99999.9999° and +99999.9999°

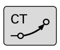
Example

```
N180 I+25  J+25*
N190 G11 G42  R+20 H+0  F250 M3*
N200 G13  H+180*
```

Circle G16 with tangential connection

The tool moves on a circular path, starting tangentially from a preceding contour element.





- **Polar coordinate radius R**: Distance between the arc end point and the pole **I, J**
- **Polar coordinate angle H**: Angular position of the arc end point.

The pole is **not** the center of the contour arc!

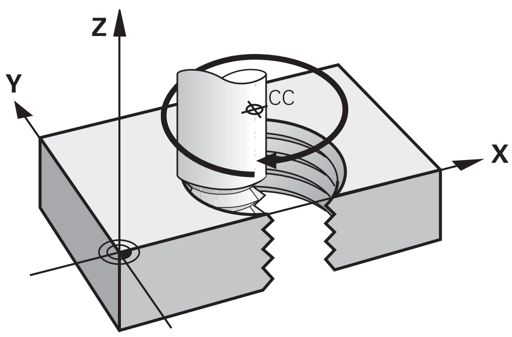
Example

```
N120 I+40  J+35*
N130 G01 G42  X+0 Y+35  F250 M3*
N140 G11  R+25 H+120*
N150 G16  R+30 H+30*
N160 G01  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

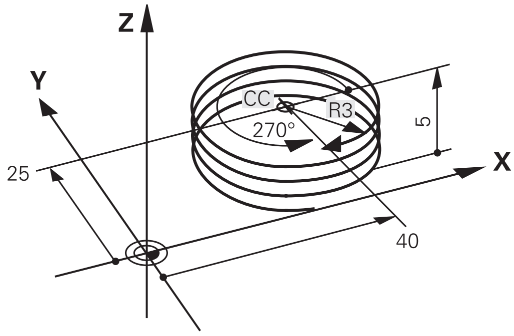
To program a helix, you must enter the total angle through which the tool is to move on the helix in incremental dimensions, and the total height of the helix.

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 **G91 H**:	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

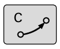
The table below illustrates in which way the shape of the helix is determined by the work direction, direction of rotation and radius compensation.

Internal thread	Work direction	Direction of rotation	Radius compensation
Right-hand Left-hand	Z+ Z+	**G13** **G12**	**G41** **G42**
Right-hand Left-hand	Z– Z–	**G12** **G13**	**G42** **G41**
External thread			
Right-hand Left-hand	Z+ Z+	**G13** **G12**	**G42** **G41**
Right-hand Left-hand	Z– Z–	**G12** **G13**	**G41** **G42**



Programming a helix

> Always enter the same algebraic sign for the direction of rotation and the incremental total angle **G91 h**. The tool may otherwise move in a wrong path and damage the contour.
>
> For the total angle **G91 h** you can enter a value of -99 999.9999° to +99 999.9999°.

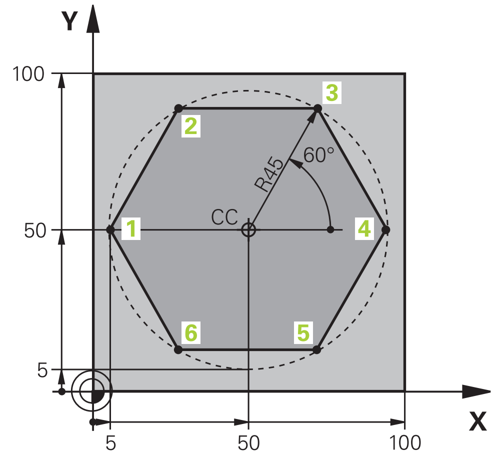


- **Polar coordinates angle:** Enter the total angle of tool traverse along the helix in incremental dimensions.
- **After entering the angle, specify the tool axis with an axis selection key**
- **Coordinate**: Enter the coordinate for the height of the helix in incremental dimensions
- **Enter the radius compensation** according to the table

Example: Thread M6 x 1 mm with 5 revolutions

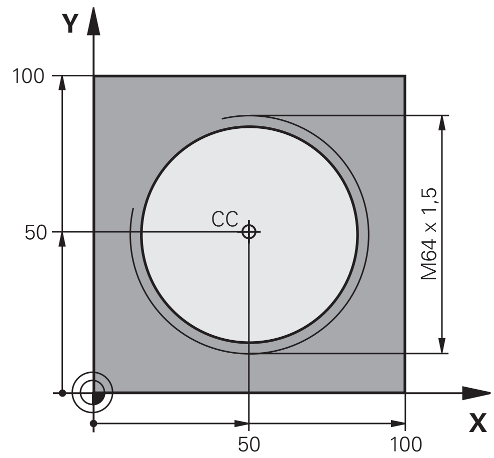
```
N120 I+40  J+25*
N130 G01  Z+0 F100  M3*
N140 G11 G41  R+3 H+270*
N150 G12 G91  H-1800 Z+5*
```

Example: Linear movement with polar coordinates



%LINEARPO G71 *	
N10 G30 G17 X+0 Y+0 Z-20*	Workpiece blank definition
N20 G31 G90 X+100 Y+100 z+0*	
N30 T1 G17 S4000*	Tool call
N40 G00 G40 G90 Z+250*	Define the preset for polar coordinates
N50 I+50 J+50*	Retract the tool
N60 G10 R+60 H+180*	Pre-position the tool
N70 G01 Z-5 F1000 M3*	Move to working depth
N80 G11 G41 R+45 H+180 F250*	Approach the contour at point 1
N90 G26 R5*	Approach the contour at point 1
N100 H+120*	Move to point 2
N110 H+60*	Move to point 3
N120 H+0*	Move to point 4
N130 H-60*	Move to point 5
N140 H-120*	Move to point 6
N150 H+180*	Move to point 1
N160 G27 R5 F500*	Tangential exit
N170 G40 R+60 H+180 F1000*	Retract the tool in the working plane, cancel radius compensation
N180 G00 Z+250 M2*	Retract in the spindle axis, end of program
N99999999 %LINEARPO G71 *	

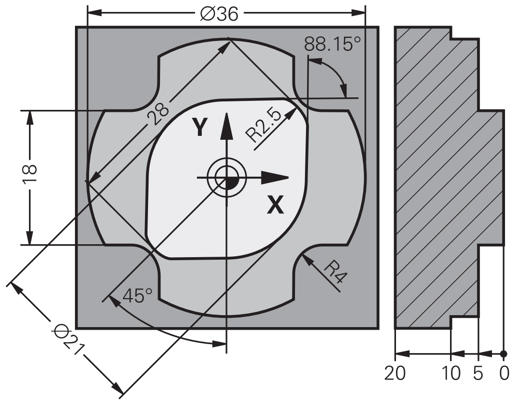
Example: Helix



%HELIX G71 *	
N10 G30 G17 X+0 Y+0 Z-20*	Workpiece blank definition
N20 G31 G90 X+100 Y+100 Z+0*	
N30 T1 G17 S1400*	Tool call
N40 G00 G40 G90 Z+250*	Retract the tool
N50 X+50 Y+50*	Pre-position the tool
N60 G29*	Transfer the last programmed position as the pole
N70 G01 Z-12,75 F1000 M3*	Move to working depth
N80 G11 G41 R+32 H+180 F250*	Approach first contour point
N90 G26 R2*	Connection
N100 G13 G91 H+3240 Z+13,5 F200*	Helical traverse
N110 G27 R2 F500*	Tangential exit
N120 G01 G40 G90 X+50 Y+50 F1000*	Retract the tool, end of program
N130 G00 Z+250 M2*	
N99999999 %HELIX G71 *	

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 dialog 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 the proximity of 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.

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 **L** command.

You cannot combine the cycle call **M89** with FK programming.

Define 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:

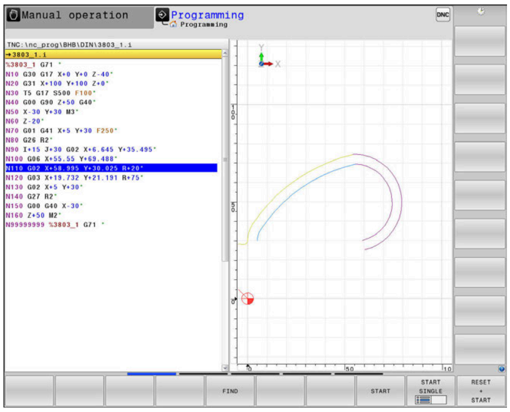
1 By the plane defined in a **FPOL** block
2 In the Z/X plane if the FK sequence is performed in turning mode
3 Through the working plane specified and defined in the **T** block (e. g. **G17** = X/Y plane)
4 If none of this applies, the standard X/Y plane is active

Display of the FK soft key depends on the spindle axis specified when defining the workpiece blank. If for example you enter spindle axis **G17** in the workpiece blank definition, the control only shows FK soft keys for the X/Y plane.

Proceed as follows if you need a working plane other than the currently active plane for programming purposes:

- ▶ Press the **PLANE XY ZX YZ** soft key
- > The control then displays the FK soft keys in the newly selected plane.

FK programming graphics



> If you wish to use graphic support during FK programming, select the **PROGRAM + GRAPHICS** screen layout.
> **Further information:** "Programming", Page 68

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 possible solutions in the standard setting

SELECT SOLUTION

- If the displayed contour element matches the drawing, select the contour element with **SELECT SOLUTION**

If you do not yet wish to select a green contour element, press the **START SINGLE** soft key to continue the FK dialog.

> Select the green contour elements as soon as possible with the **SELECT SOLUTION** soft key. This way you can reduce the ambiguity of subsequent 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 OMIT BLOCK NR.** soft key to **SHOW** (soft-key row 3)

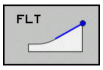
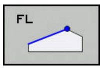
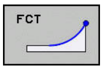
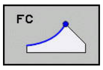
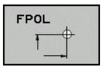
Initiating the FK dialog

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
	Straight line with tangential connection
	Straight line without tangential connection
	Circular arc with tangential connection
	Circular arc without tangential connection
	Pole for FK programming
	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:

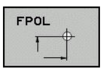
FK

- ▶ Press the **FK** key again

Pole for FK programming

FK

- ▶ To display the soft keys for free contour programming, press the **FK** key



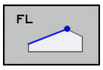
- ▶ To initiate the dialog for defining the pole, press the **FPOL** soft key
- > The control displays the axis soft keys of the active working plane.
- ▶ Enter the pole coordinates using these soft keys

> **i** 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

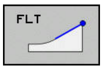


- To initiate the dialog for free programming of straight lines, press the **FL** soft key
- > The control displays additional soft keys.
- Enter all known data in the NC block by using these soft keys
- > The FK graphic displays the programmed contour element in violet until sufficient data is entered. If the entered data describes several solutions, the graphic will display the contour element in green.
  Further information: "FK programming graphics", Page 169

Straight line with tangential connection

If the straight line connects tangentially to another contour element, initiate the dialog with the soft key:

- To display the soft keys for free contour programming, press the **FK** key

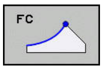


- To initiate the dialog, press the **FLT** soft key
- Enter all known data in the NC block by using the soft keys

Free circular path programming

Circular arc without tangential connection

- To display the soft keys for free contour programming, press the **FK** key

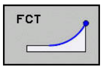


- To initiate the dialog for free programming of circular arcs, press the **FC** soft key
- > The control displays soft keys with which you can enter direct data on the circular arc or data on the circle center.
- Enter all known data in the NC block by using these soft keys
- > The FK graphic displays 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 169

Circular arc with tangential connection

If the circular arc connects tangentially to another contour element, initiate the dialog with the **FCT** soft key:

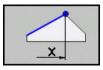
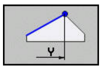
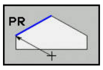
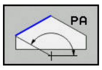
- To display the soft keys for free contour programming, press the **FK** key

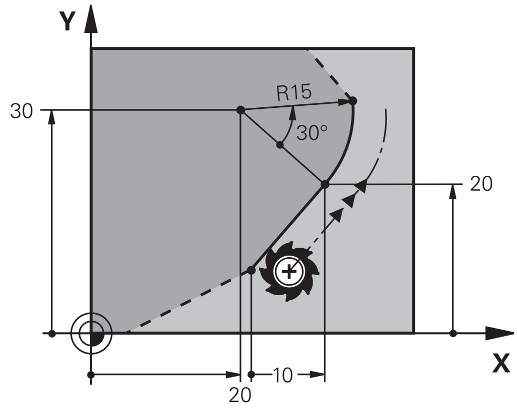


- To initiate the dialog, press the **FCT** soft key
- Enter all known data in the NC block by using the soft keys

Input possibilities

End point coordinates

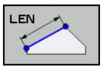
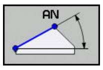
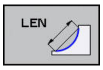
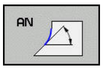
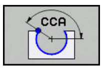
Soft keys		Known data
		Cartesian coordinates X and Y
		Polar coordinates referenced to FPOL

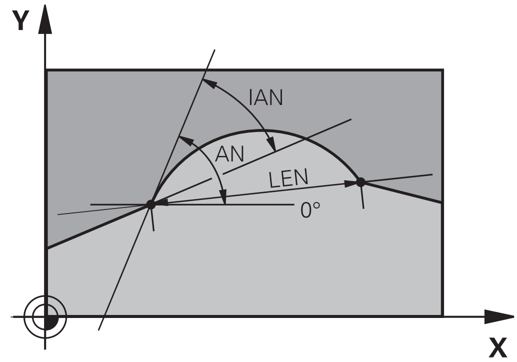


Example

```
N70 FPOL  X+20 Y+30*
N80 FL  IX+10 Y+20  G42 F100*
N90 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

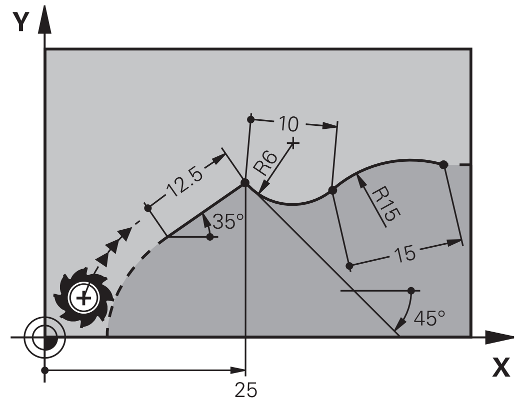


NOTICE

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



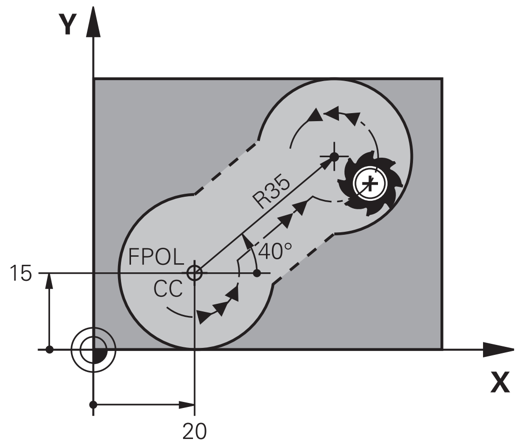
Example

```
N20 FLT  X+25 LEN 12.5  AN+35 G41  F200*
N30 FC  DR+ R6  LEN 10 AN-45*
N40 FCT  DR- R15  LEN 15*
```

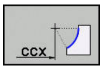
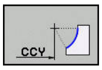
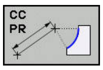
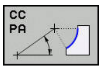
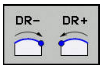
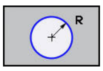
Circle center CC, radius and direction of rotation in the FC/FCT block

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 coordi-nates
		Center point in polar coordinates
		Rotational direction of the arc
		Radius of an arc

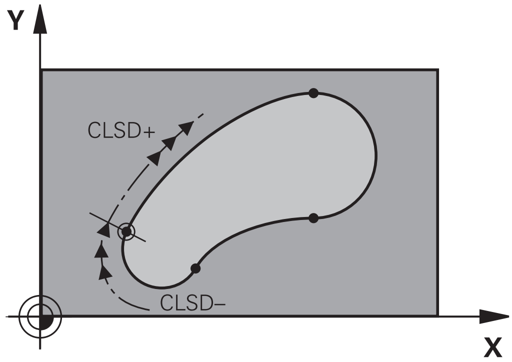
Example

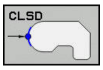
```
N10 FC  CCX+20 CCY+15  DR+ R15*
N20 FPOL  X+20 Y+15*
N30 FL  AN+40*
N40 FC  DR+ R15  CCPR+35 CCPA+40*
```

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

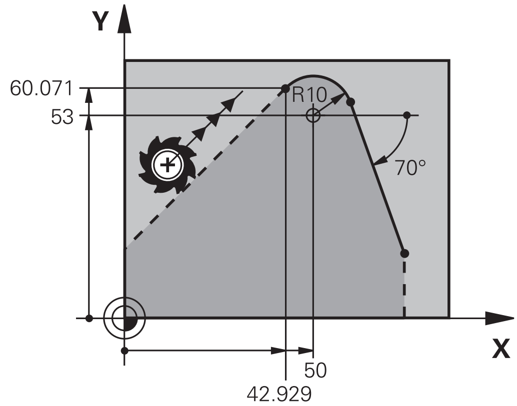
```
N10 G01  X+5 Y+35  G41 F500  M3*
N20 FC  DR- R15  CLSD+ CCX+20  CCY+35*
...
N30 FCT  DR- R+15  CLSD-*
```

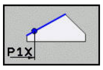
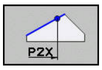
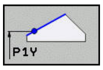
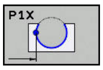
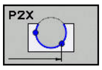
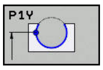
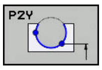
Auxiliary points

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.

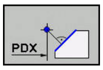
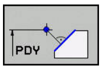
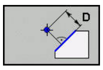
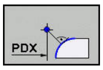
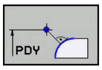
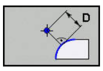
Auxiliary points on a contour

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

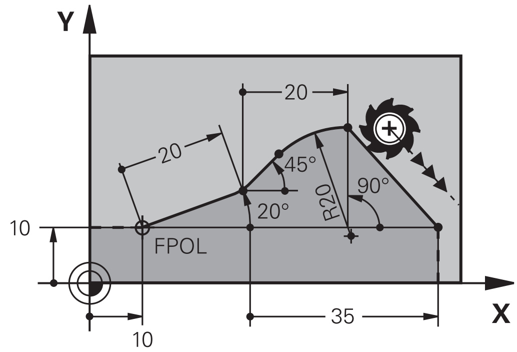
Soft keys		Known data
		X and Y coordinates of the auxiliary point near a straight line
		Distance of auxiliary point to straight line
		X and Y coordinates of an auxiliary point near a circular arc
		Distance of auxiliary point to circular arc

Example

```
N10 FC  DR- R10  P1X+42.929 P1Y+60.071*
N20 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

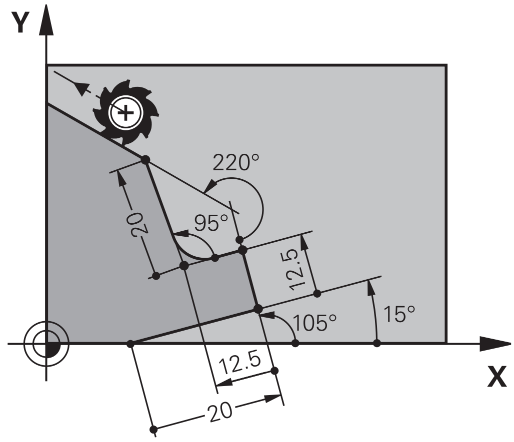
Soft keys		Known data
RX N...	RY N...	Cartesian coordinates relative to NC block N
RPR N...	RPA N...	Polar coordinates relative to NC block N

Example

```
N10 FPOL  X+10 Y+10*
N20 FL  PR+20 PA+20*
N30 FL  AN+45*
N40 FCT  IX+20 DR-  R20 CCA+90  RX 20*
N50 FL  IPR+35 PA+0  RPR 20*
```

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

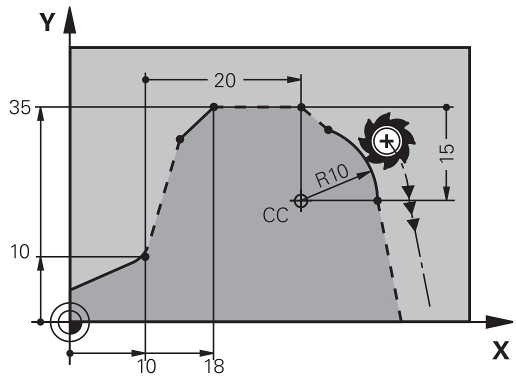


Example

```
N10 FL  LEN 20 AN+15*
N20 FL  AN+105 LEN 12.5*
N30 FL  PAR 10  DP 12.5*
N40 FSELECT 2*
N50 FL  LEN 20 IAN+95*
N60 FL  IAN+220 RAN 20*
```

Data relative to NC block N: Circle center CC

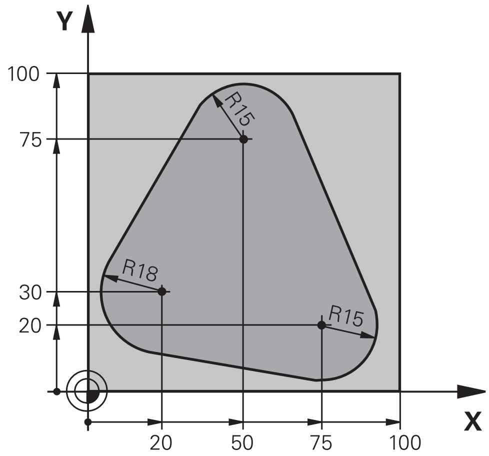
Soft key		Known data
RCCX N...	RCCY N...	Cartesian coordinates of the circle center relative to NC block N
RCCPR N...	RCCPA N...	Polar coordinates of the circle center relative to NC block N



Example

```
N10 FL  X+10 Y+10  G41*
N20 FL  ...*
N30 FL  X+18 Y+35*
N40 FL  ...*
N50 FL  ...*
N60 FC  DR- R10  CCA+0 ICCX+20  ICCY-15 RCCX10  RCCY30*
```

Example: FK programming 1



%FK1 G71 *	
N10 G30 G17 X+0 Y+0 Z-20*	Workpiece blank definition
N20 G31 X+100 Y+100 Z+0*	
N30 T 1 G17 S500*	Tool call
N40 G00 G90 Z+250 G40 M3*	Retract the tool
N50 G00 X-20 Y+30 G40*	Pre-positioning the tool
N60 G01 Z-10 G40 F1000*	Move to working depth
N70 APPR CT X+2 Y+30 CCA90 R+5 G41 F250*	Approach the contour on a circular arc with tangential connection
N80 FC DR- R18 CLSD+ CCX+20 CCY+30*	FK contour section:
N90 FLT*	Program all known data for each contour element
N100 FCT DR- R15 CCX+50 CCY+75*	
N110 FLT*	
N120 FCT DR- R15 CCX+75 CCY+20*	
N130 FLT*	
N140 FCT DR- R18 CLSD- CCX+20 CCY+30*	
N150 DEP CT CCA90 R+5 F2000*	Depart the contour on a circular arc with tangential connection
N160 G00 X-30 Y+0*	
N170 G00 Z+250 M2*	Retract the tool, end of program
N99999999 %FK1 G71 *	

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
BLOCK N	Jump to the block number entered
BLOCK N	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

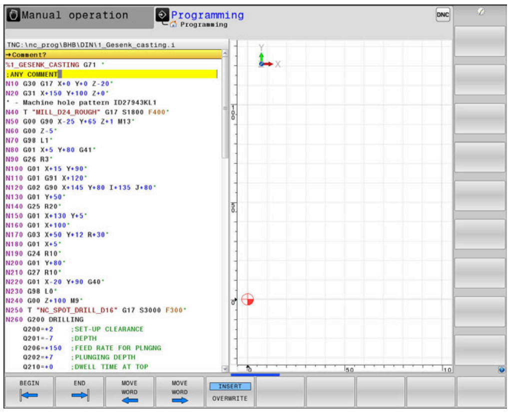
Opening the selection window with the GOTO key

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

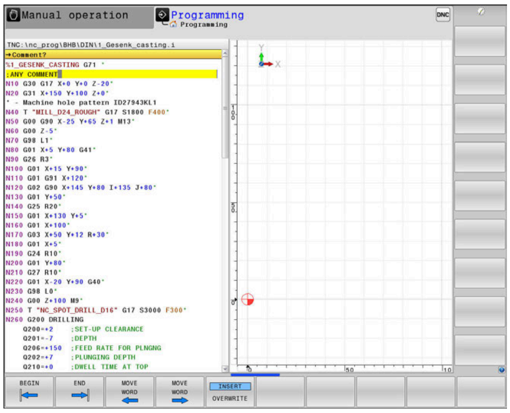
Scrollbar

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

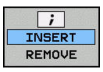
Entering a comment in a separate NC block

- 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

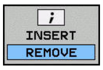
Alternative:

- 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

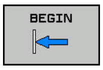
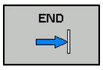
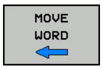
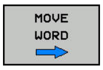
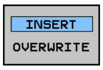


- Press the **REMOVE COMMENT** soft key

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 of the 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.
- ▶ Select the **TEXT EDITOR** option

OK

- ▶ 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.

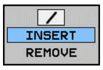
6.5 Skipping NC blocks

Insert a slash (/)

You can optionally hide NC blocks.

Proceed as follows to hide NC blocks in the **Programming** operating mode:

- Select the desired NC block

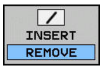


- Press the **INSERT** soft key
- > The control inserts a slash (/).

Delete the slash (/)

Proceed as follows to show NC blocks in the **Programming** operating mode again:

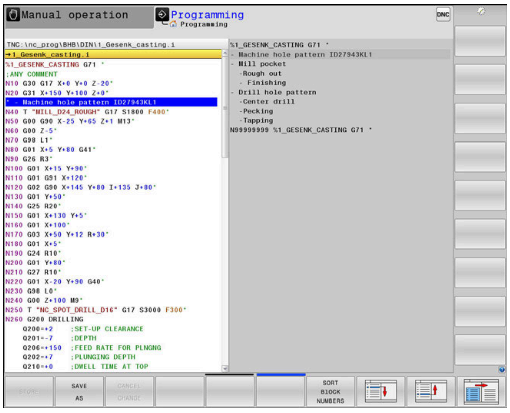
- Select the hidden NC block



- 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.

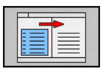
The control manages the inserted structure items in a separate file (extension: .SEC.DEP). This speeds navigation in the program structure window.

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

SPEC FCT
- Press the **SPEC FCT** key

PROGRAM-MING AIDS
- Press the **PROGRAMMING AIDS** soft key

INSERT SECTION
- Press the **INSERT SECTION** soft key
- Enter the structuring text
- Change the structuring depth (indenting) via soft key

> The structure items can be indented only during editing.

> You can also insert structure blocks with the key combination **Shift + 8**.

Selecting blocks in the program structure window

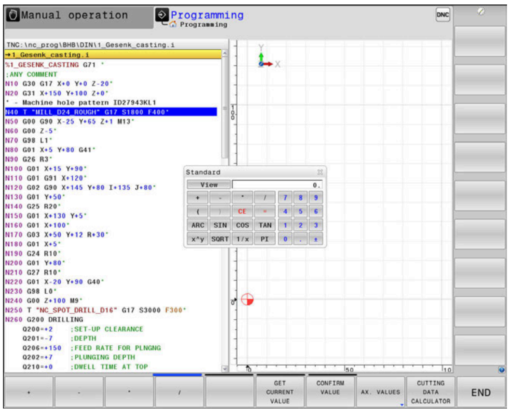
If you are scrolling through the program structure window block by block, the control at the same time automatically moves the corresponding NC blocks in the program window. This way you can quickly skip large program sections.

6.7 Calculator

Operation

The control features an integrated calculator with the basic mathematical functions.

- Press the **CALC** key to show the calculator
- Select the arithmetical functions: The calculator is operated with short commands via soft key or through an alphabetic keyboard
- Press the **CALC** key to close the calculator



Calculate function	Shortcut (soft key)
Addition	+
Subtraction	–
Multiplication	*
Division	/
Calculating with parentheses	( )
Arc cosine	ARC
Sine	SIN
Cosine	COS
Tangent	TAN
Powers of values	X^Y
Square root	SQRT
Inversion	1/x
pi (3.14159265359)	PI
Add value to buffer memory	M+
Save the value to buffer memory	MS
Recall from buffer memory	MR
Delete buffer memory contents	MC
Natural logarithm	LN
Logarithm	LOG
Exponential function	e^x
Check the algebraic sign	SGN
Form the absolute value	ABS

Calculate function	Shortcut (soft key)
Truncate decimal places	INT
Truncate places before the decimal point	FRAC
Modulus operator	MOD
Select view	View
Delete value	CE
Unit of measure	MM or INCH
Show angle values in radians (standard: angle in degrees)	RAD
Select the display mode of the numerical value	DEC (decimal) or HEX (hexadecimal)

Transferring the calculated value into the NC program

- Use the arrow keys to select the word into which the calculated value is to be transferred
- Superimpose the on-line 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

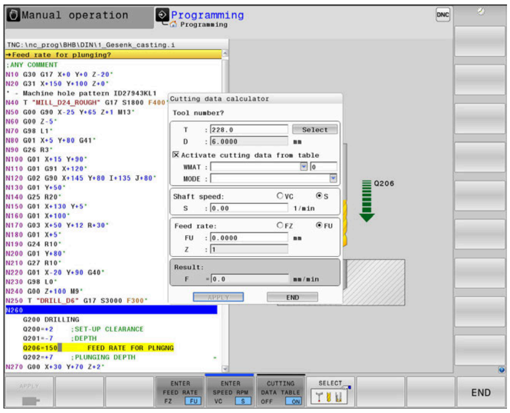
Soft key	Function
AX. VALUES	Load the nominal or reference value of the respective axis position into the calculator
GET CURRENT VALUE	Load the numerical value from the active input field into the calculator
CONFIRM VALUE	Load the numerical value from the calculator field into the active input field
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
- open the dialog field for spindle speed input in the T block
- open the dialog field for feed rate input in positioning blocks or cycles
- press the **F** soft key in the **Manual Operation** mode
- press the **S** soft key in the **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.

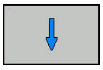
Window for feed rate calculation:

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 **T** 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 **T** 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

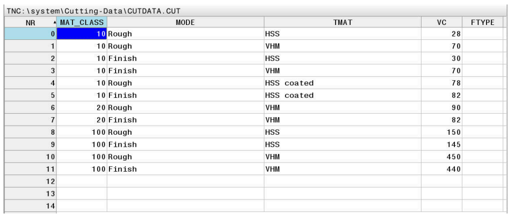
Cutting materials are defined in the TMAT.tab table. You must save this table in the directory **TNC:\table**.

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

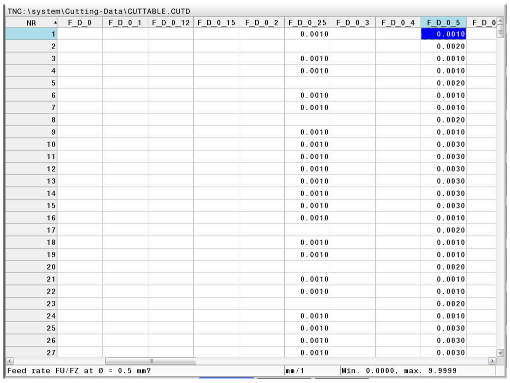
Diameter-dependent cutting data table

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	

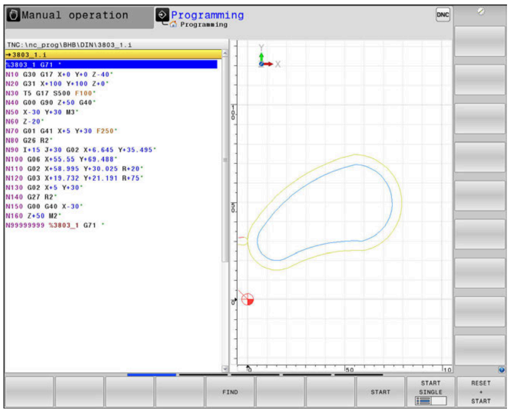
Feed rate FU/FZ at Ø = 0.5 mm? | mm/1 | Min. 0.0000, max. 9.9999

> 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.

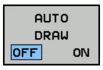
6.9 Programming graphics

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 wish to have the control generate graphics during programming, set the **AUTO DRAW** soft key to **OFF**.

> If **AUTO DRAW** is set to **ON**, 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 if 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 169

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 + START

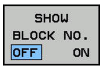
- Reset previously active tool data and generate graphics: Press the **RESET + START** soft key

Additional functions:

Soft key	Function
RESET + START	Reset previously active tool data. Generate programming graphics
START SINGLE	Generate programming graphic blockwise
START	Generate a complete graphic or complete it after **RESET + START**
STOP	Stop the programming graphics. This soft key only appears while the control is generating the programming graphics
VIEWS	Selecting views ■ Plan view ■ Front view ■ Page view
TOOL PATH: SHOW HIDE	Display or hide tool paths
SHOW FMAX PATHS OFF ON	Display or hide tool paths in rapid traverse

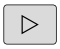
Block number display ON/OFF

- Shift the soft-key row



- Display block numbers: **BLOCK NO.** soft keySet **BLOCK NO. SHOW OMIT** to **SHOW**
- Hide block numbers: **BLOCK NO.** soft keySet **BLOCK NO. SHOW OMIT** to **HIDE**

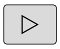
Erasing the graphic



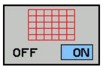
- Shift the soft-key row

- Erase the graphics: Press the **CLEAR GRAPHICS** soft key

Showing grid lines

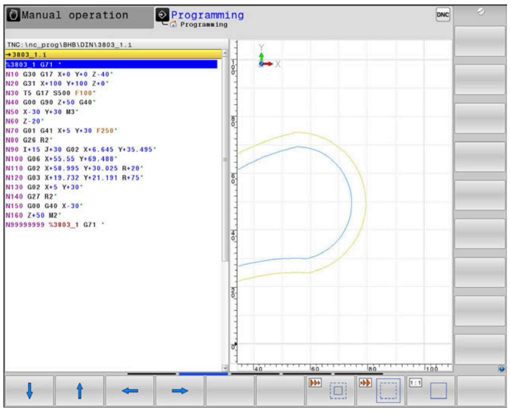


- Shift the soft-key row



- Show grid lines: Press the **Show grid lines** soft key

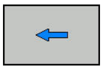
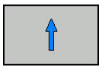
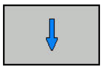
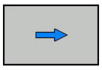
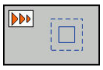
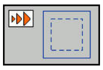
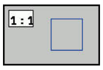
Magnification or reduction of details



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

With the **RESET BLK FORM** soft key, you can restore the original section.

You can also use the mouse to change the graphic display. The following functions are available:

- To shift the model, hold the center mouse button or mouse wheel down and move the mouse. If you simultaneously press the shift key, you can only shift the model horizontally or vertically.
- To zoom in on a certain area, mark a zoom area by holding the left mouse button down. After you release the left mouse button, the control zooms in on the defined area.
- To rapidly magnify or reduce any area, rotate the mouse wheel backwards or forwards.

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

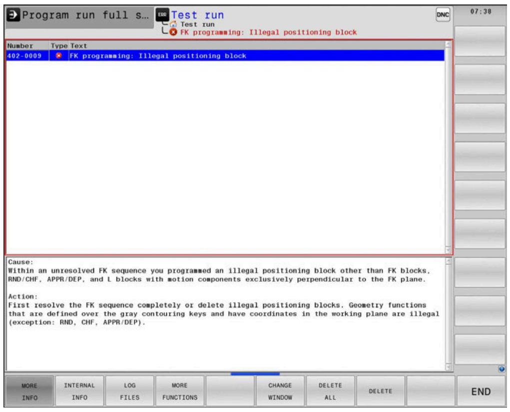
- ▶ Press the **END** soft key, or

ERR

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

Detailed error messages

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



- Open the error window

MORE INFO

- Information on the error cause and corrective action: Position the cursor on the error message and press the **MORE INFO** soft key
- > The control opens a window with information on the error cause and corrective action.
- Leave Info: Press the **MORE INFO** soft key again

Soft key: INTERNAL INFO

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

- Open the error window

INTERNAL INFO

- Detailed information about the error message: Position the cursor on the error message and press the **INTERNAL INFO** soft key
- > The control opens a window with internal information about the error.
- Exit the details: Press the **INTERNAL INFO** soft key again

Soft key FILTER

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

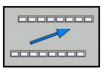
- Open the error window

MORE FUNCTIONS

- Press the **MORE FUNCTIONS** soft key

FILTER OFF ON

- Press the **FILTER** soft key. The control filters the identical warnings



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

Clearing errors

Clearing errors outside of the error window

CE

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

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

Clearing errors

- Open the error window

DELETE

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

DELETE ALL

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

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

Error log

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

- Open the error window.

LOG FILES

- Press the **LOG FILES** soft key

ERROR LOG

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

PREVIOUS FILE

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

CURRENT FILE

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

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

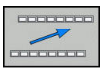
Keystroke log

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

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

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

Overview of the keys and soft keys for viewing the log

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

Informational texts

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

Saving service files

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

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

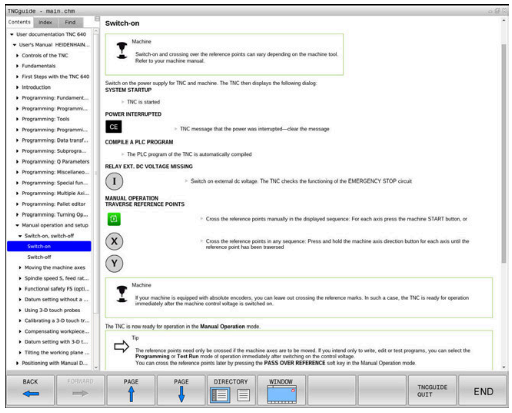
MACHINE MFR.

- 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 212



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, with the contents of all existing .chm files.

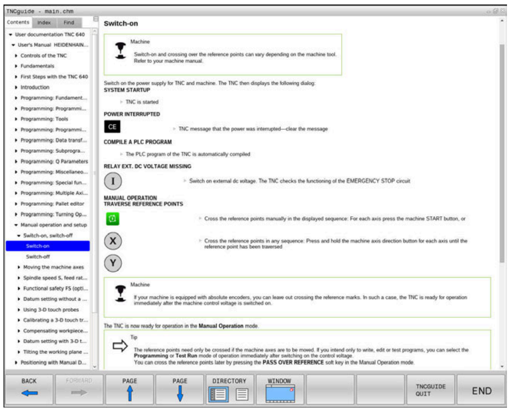
> As an option, your machine tool builder can embed machine-specific documentation in the **TNCguide**. These documents then appear as a separate book in the **main.chm** file.

Working with TNCguide

Calling TNCguide

There are several ways to start the TNCguide:

- Press the **HELP** key.
- Click the help symbol at the lower right of the screen beforehand, then click the appropriate soft keys
- Open a help file (CHM file) via the file management. The control can open any .chm file, even if it is not saved in the control's internal memory



> On the Windows programming station, the TNCguide is opened in the internally defined standard browser.

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

- Select the soft-key row containing the desired soft key
- Click with the mouse on the help symbol that the control displays just above the soft-key row
- > The mouse pointer turns into a question mark.
- Move the question mark to the soft key for which you want an explanation
- > The control opens the TNCguide. If there is no entry point for the selected soft key, then the control opens the book file **main.chm**. You can search for the desired explanation using full text search or by using the navigation.

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

- Select any NC block
- Select the desired word
- Press the **HELP** key.
- > The control opens the Help system and shows the description of the active function. This does not apply for miscellaneous functions or cycles from your machine manufacturer.

Navigating in the TNCguide

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

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

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

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

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

Subject index

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

The left side is active.

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

Alternative:

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

Full-text search

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

The left side is active.

- Select the **Find** tab
- Activate the **Find:** entry field
- Enter the search word
- Press the **ENT** key
- > The control lists all sources containing the word.
- Use the arrow keys to navigate to the desired source
- Press the **ENT** key to go to the selected source

> The full-text search only works for single words.
> If you activate the **Search only in titles** function, the control searches only through headings and ignores the body text. To activate the function, use the mouse or select it and then press the space bar to confirm.

Downloading current help files

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

Navigate to the suitable help file as follows:

- TNC Controls
- Series, e.g. TNC 600
- Desired NC software number, e.g.TNC 640 (34059x-09)
- 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. Some miscellaneous functions can be programmed with additional parameters. In this case, the dialog is continued for the parameter input.

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**

Example

```
N87 G38  M6*
```

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.

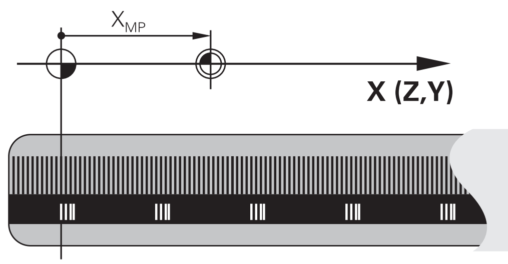
M	Effect	Effective at block	Start	End
M0	Program STOP Spindle STOP			■
M1	Optional program STOP Spindle STOP if necessary Coolant OFF if necessary (function defined by the machine tool builder)			■
M2	STOP program run Spindle STOP Coolant off Return jump to block 1 Clear status display Functional scope 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			■
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 (e.g. 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 manufacturer 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 be based on 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 screen reference 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.

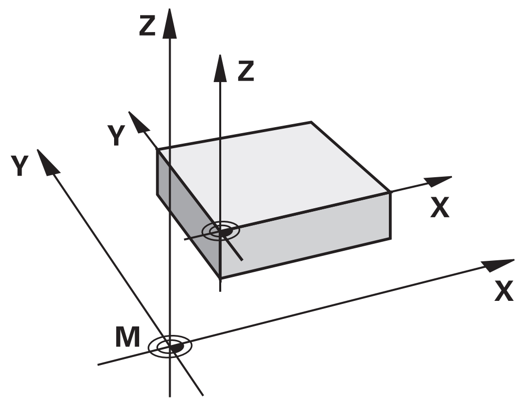
M91 and M92 take effect at the start of block.

Workpiece preset

If you want the coordinates to always be referenced to the machine datum, you can disable the setting of presets 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.

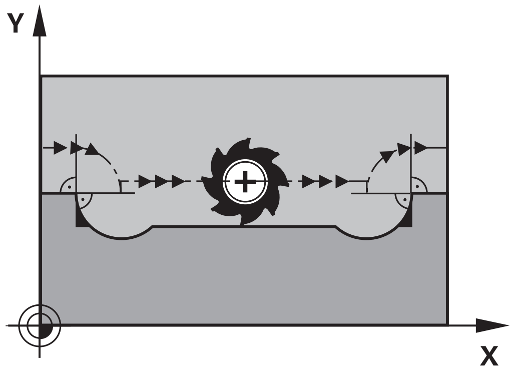
7.4 Miscellaneous functions for path behavior

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.

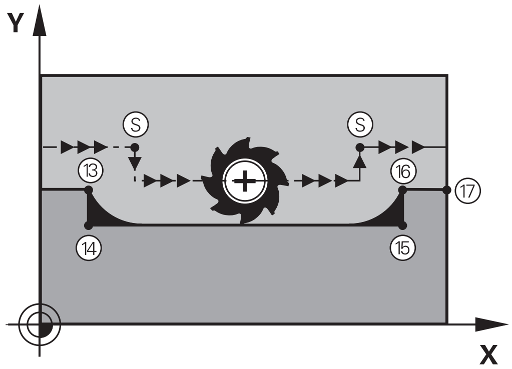
In such cases, the control interrupts the program run and generates the **Tool radius too large** error message.



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 224

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

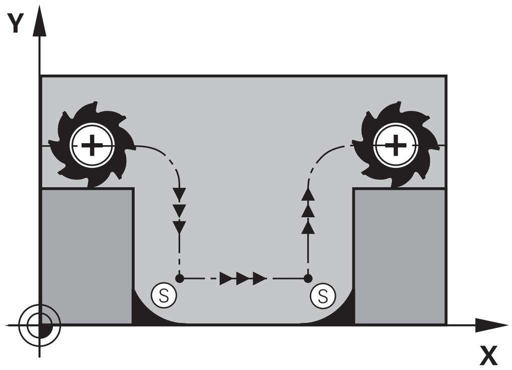
N50 G99 G01 ... R+20*	Large tool radius
...	
N130 X ... Y ... F ... M97*	Move to contour point 13
N140 G91 Y-0.5 ... F ...*	Machine small contour step 13 to 14
N150 X+100 ...*	Move to contour point 15
N160 Y+0.5 ... F ... M97*	Machine small contour step 15 to 16
N170 G90 X ... Y ... *	Move to contour point 17

Machining open contour corners: M98

Standard behavior

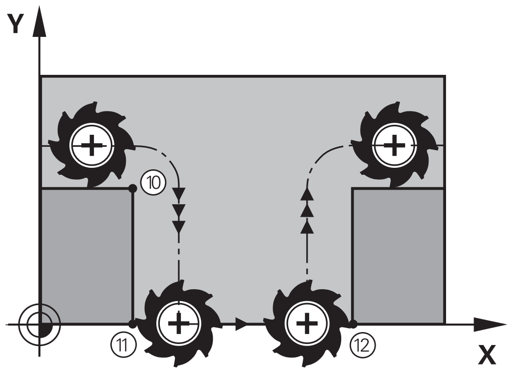
The control calculates the intersections of the cutter paths at inside corners and moves the tool in the new direction at those points.

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

N100 G01 G41 X ... Y ... F ...*
N110 X ... G91 Y ... M98*
N120 X+ ...*

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):
N170 G01 G41 X+20 Y+20 F500 M103 F20*	500
N180 Y+50*	500
N190 G91 Z-2.5*	100
N200 Y+5 Z-5*	141
N210 X+50*	500
N220 G90 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

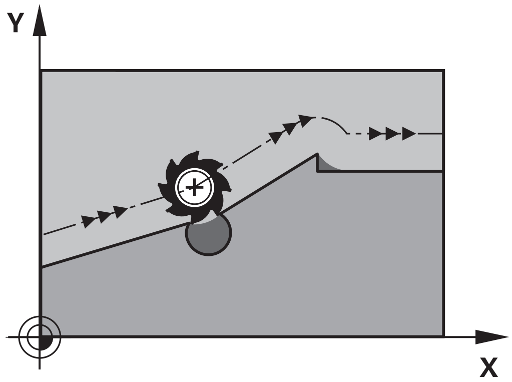
If the tool radius is larger than the contour step that needs to be machined with radius compensation, the control interrupts program run and generates an error message. **M97** inhibits the error message, but this results in dwell marks and will also move the corner.

Further information: "Machining small contour steps: M97", Page 220

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

M120 must be included in an NC block that also contains an **G41** or **G42** radius compensation. **M120** is then effective from this NC block until you

- radius compensation is canceled with **G40**
- **M120 LA0** is programmed
- **M120** is programmed without **LA**
- call another NC program with **%**
- the working plane is tilted with Cycle **G80** or with the **PLANE** function

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 **G60** Tolerance
  - Cycle **G80** Working plane
  - **PLANE** function
  - **M114**
  - **M128**

Superimposing handwheel positioning during program run: M118

Standard behavior

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.
>
> The **M118** handwheel superimpositioning function cannot be used in combination with the **Dynamic Collision Monitoring (DCM)** function and the additional **TCPM** or **M128** function.
>
> 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).

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.

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.

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:

```
N250 G01  G41 X+0  Y+38.5 F125  M118 X1  Y1 B5*
```

> **M118** is always effective in the machine coordinate system.
>
> If the Global Program Settings option (option 44) is active, **M118** is in effect in the coordinate system selected most recently for handwheel superimpositioning. To view the coordinate system active for **M118**, press the **3D-ROT** soft key.
>
> **Further information:** User's Manual for Setup, Testing and Running NC Programs
>
> **M118** is also effective in the **Positioning w/ Manual Data Input** operating mode!

Virtual tool axis VT

> 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 a 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 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

```
N250 G01  X+0 Y+38.5  F125 M140  MB50*
N251 G01  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 the **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

```
G01  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 **(G98 L)** labels.

A LABEL is identified by a number between 1 and 65535 or by a name you define. Each LABEL number or LABEL name can be set only once in the NC program with the **LABEL SET** key or by entering **G98**. The number of label names you can enter is only limited by the internal memory.

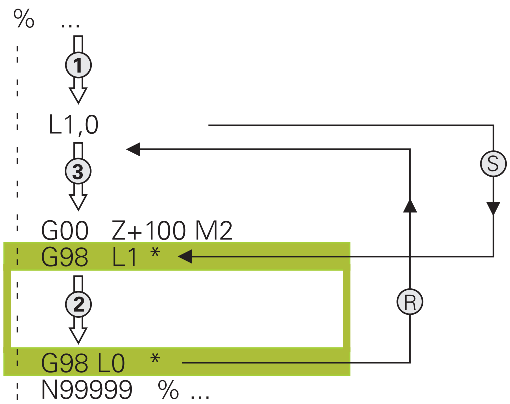
> Do not use a label number or label name more than once!

Label 0 (**G98 L0**) is used exclusively to mark the end of a subprogram and can therefore be used as often as desired.

8.2 Subprograms

Operating sequence

1. The control executes the NC program up to the block in which a subprogram is called with **Ln,0**
2. The subprogram is then executed until the subprogram end **G98 L0**
3. The control then resumes the NC program from the NC block after the subprogram call **Ln,0**



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

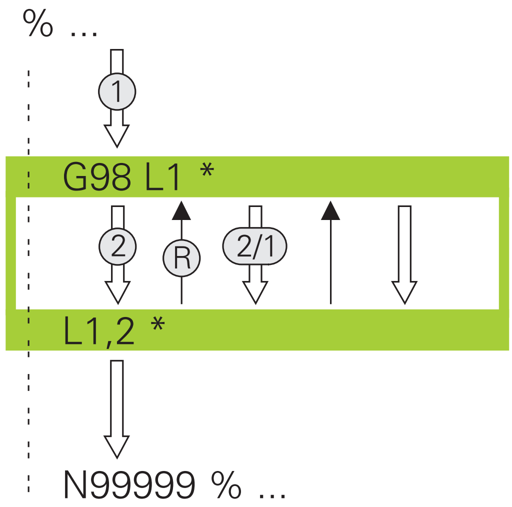
- Call a subprogram: Press the **LBL CALL** key
- Enter the subprogram number of the subprogram you wish to call. If you want to use a label name, press the **LBL NAME** soft key to switch to text entry.

L 0 is not permitted (Label 0 is only used to mark the end of a subprogram).

8.3 Program-section repeats

Label G98

The beginning of a program section repeat is marked by the label **G98 L**. The end of a program section repeat is identified by **Ln,m**.



Operating sequence

1 The control executes the NC program up to the end of the program section (**Ln,m**)
2 Then the program section between the called LABEL and the label call **Ln,m** is repeated the number of times entered after **m**
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 Any desired NC program as subprogram

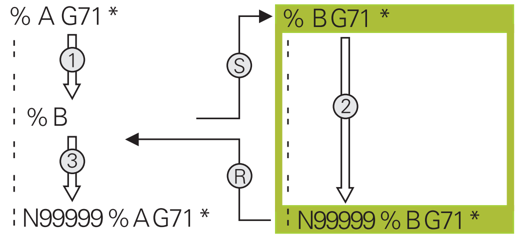
Overview of the soft keys

When you press the **PGM CALL** key, the control displays the following soft keys:

Soft key	Function
CALL PROGRAM	Call an NC program with **%**
SELECT DATUM TABLE	Select a datum table with **%:TAB:**
SELECT POINT TABLE	Select a point table with **%:PAT:**
SELECT CONTOUR	Select a contour program with **%:CNT:**
SELECT PROGRAM	Select an NC program with **%:PGM:**
CALL SELECTED PROGRAM	Call the last selected file with **%<>%**
SELECT CYCLE	Select any NC program with **G: :** 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 **%**.
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.



Programming notes

- The control does not require any labels to call any part program
- The called NC program must not contain any **%** call into the calling NC program (an endless loop ensues)
- The called NC program must not contain the miscellaneous functions **M2** or **M30**. If you have defined subprograms with labels in the called NC program, you can then replace M2 or M30 with the **D09 P01 +0 P02 +0 P03 99** jump function
- 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 **G39**.
- You can also call any NC program with the function **Select the cycle** (**G: :**).
- As a rule, Q parameters are effective globally with a program call with **%**. So please note that changes to Q parameters in the called NC program can also influence the calling NC program.

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 **N99999999** 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 NC program as a subprogram

Calling a program with Calling a program

The **%** function calls any NC program as a subprogram. The control runs the called 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 SELECT PROGRAM and CALL SELECTED PROGRAM

Use the function **%:PGM:** to select any NC program as a subprogram and call it at another position in the NC program. The control runs the called NC program from the position where you called it with **%<>%** in the NC program.

The **%: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 **%<>%** to call the NC program that was selected last.

> If an NC program that was called using **%<>%** 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 **D18** (**ID10 NR110** and **NR111**) to check all paths at the beginning of the program.
> **Further information:** "D18 – Reading system data", Page 282

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

Nesting depth

The nesting depth is the number of successive levels in which program sections or subprograms can call further program sections or subprograms.

- Maximum nesting depth for subprograms: 19
- Maximum nesting depth for main program calls: 19, where a **G79** acts like a main program call
- You can nest program section repeats as often as desired

Subprogram within a subprogram

Example

%UPGMS G71 *	
...	
N17 L "UP1",0*	Subprogram at label G98 L1 is called
...	
N35 G00 G40 Z+100 M2*	Last program block of the
	main program with M2
N36 G98 L "UP1"	Beginning of subprogram SP1
...	
N39 L2,0*	Subprogram at label G98 L2 is called
...	
N45 G98 L0*	End of subprogram 1
N46 G98 L2*	Beginning of subprogram 2
...	
N62 G98 L0*	End of subprogram 2
N99999999 %UPGMS G71 *	

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

%REPS G71 *	
...	
N15 G98 L1*	Beginning of program section repeat 1
...	
N20 G98 L2*	Beginning of program section repeat 2
...	
N27 L2,2*	Program section call with two repeats
...	
N35 L1,1*	The program section between this NC block and G98 L1
...	(NC block 15) is repeated once
N99999999 %REPS G71 *	

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

%UPGREP G71 *	
...	
N10 G98 L1*	Beginning of program section repeat 1
N11 L2,0*	Subprogram call
N12 L1,2*	Program section call with two repeats
...	
N19 G00 G40 Z+100 M2*	Last NC block of the main program with M2
N20 G98 L2*	Beginning of subprogram
...	
N28 G98 L0*	End of subprogram
N99999999 %UPGREP G71 *	

Program execution

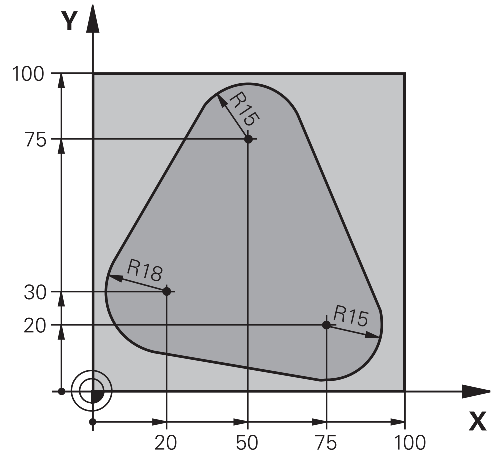
1. Main program UPGREP is executed up to NC block 11
2. Subprogram 2 is called and executed.
3. The program section between NC block 12 and NC block 10 is repeated twice. This means that subprogram 2 is repeated twice
4. Main program UPGREP is executed from NC block 13 up to NC block 19. Return jump 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

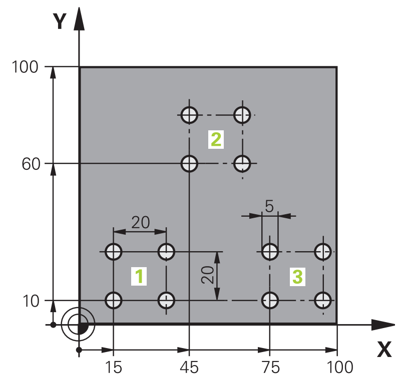


%PGMREP G71 *	
N10 G30 G17 X+0 Y+0 Z-40*	
N20 G31 G90 X+100 Y+100 Z+0*	
N30 T1 G17 S3500*	Tool call
N40 G00 G40 G90 Z+250*	Retract the tool
N50 I+50 J+50*	Set pole
N60 G10 R+60 H+180*	Pre-position in the working plane
N70 G01 Z+0 F1000 M3*	Pre-position to the workpiece surface
N80 G98 L1*	Set label for program section repeat
N90 G91 Z-4*	Infeed depth in incremental values (in space)
N100 G11 G41 G90 R+45 H+180 F250*	First contour point
N110 G26 R5*	Contour approach
N120 H+120*	
N130 H+60*	
N140 H+0*	
N150 H-60*	
N160 H-120*	
N170 H+180*	
N180 G27 R5 F500*	Contour departure
N190 G40 R+60 H+180 F1000*	Retract tool
N200 L1,4*	Return jump to label 1; section is repeated a total of 4 times
N200 G00 Z+250 M2*	Retract the tool, end of program
N99999999 %PGMWDH G71 *	

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

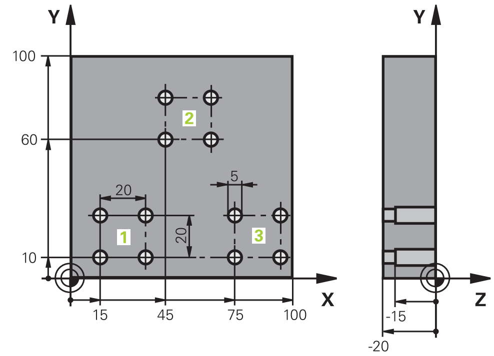


%SP1 G71 *	
N10 G30 G17 X+0 Y+0 Z-40*	
N20 G31 G90 X+100 Y+100 Z+0*	
N30 T1 G17 S3500*	Tool call
N40 G00 G40 G90 Z+250*	Retract the tool
N50 G200 DRILLING	Define the DRILLING cycle
Q200=2 ;SET-UP CLEARANCE	
Q201=-30 ;DEPTH	
Q206=300 ;FEED RATE FOR PLNGNG	
Q202=5 ;PLUNGING DEPTH	
Q210=0 ;DWELL TIME AT TOP	
Q203=+0 ;SURFACE COORDINATE	
Q204=2 ;2ND SET-UP CLEARANCE	
Q211=0 ;DWELL TIME AT DEPTH	
Q395=0 ;DEPTH REFERENCE	
N60 X+15 Y+10 M3*	Move to starting point for group 1
N70 L1,0*	Call the subprogram for the group
N80 X+45 Y+60*	Move to starting point for group 2
N90 L1,0*	Call the subprogram for the group
N100 X+75 Y+10*	Move to starting point for group 3
N110 L1,0*	Call the subprogram for the group
N120 G00 Z+250 M2*	End of main program
N130 G98 L1*	Beginning of subprogram 1: Group of holes
N140 G79*	Call cycle for 1st hole
N150 G91 X+20 M99*	Move to 2nd hole, call cycle
N160 Y+20 M99*	Move to 3rd hole, call cycle
N170 X-20 G90 M99*	Move to 4th hole, call cycle
N180 G98 L0*	End of subprogram 1
N99999999 %UP1 G71 *	

Example: Group of holes with several 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



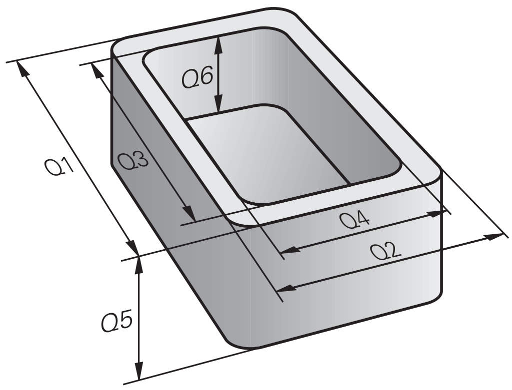
%SP2 G71 *	
N10 G30 G17 X+0 Y+0 Z-40*	
N20 G31 G90 X+100 Y+100 Z+0*	
N30 T1 G17 S5000*	Centering drill tool call
N40 G00 G40 G90 Z+250*	Retract the tool
N50 G200 DRILLING	Define the CENTERING cycle
Q200=2 ;SET-UP CLEARANCE	
Q201=-3 ;DEPTH	
Q206=250 ;FEED RATE FOR PLNGNG	
Q202=3 ;PLUNGING DEPTH	
Q210=0 ;DWELL TIME AT TOP	
Q203=+0 ;SURFACE COORDINATE	
Q204=10 ;2ND SET-UP CLEARANCE	
Q211=0.2 ;DWELL TIME AT DEPTH	
Q395=0 ;DEPTH REFERENCE	
N60 L1,0*	Call subprogram 1 for the entire hole pattern
N70 G00 Z+250 M6*	Tool change
N80 T2 G17 S4000*	Drill tool call
N90 D0 Q201 P01 -25*	New depth for drilling
N100 D0 Q202 P01 +5*	New plunging depth for drilling
N110 L1,0*	Call subprogram 1 for the entire hole pattern
N120 G00 Z+250 M6*	Tool change
N130 T3 G17 S500*	Reamer tool call
N140 G201 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	
N150 L1,0*	Call subprogram 1 for the entire hole pattern

N160 G00 Z+250 M2*	End of main program
N170 G98 L1*	Beginning of subprogram 1: Entire hole pattern
N180 G00 G40 G90 X+15 Y+10 M3*	Move to starting point for group 1
N190 L2,0*	Call subprogram 2 for the group
N200 X+45 Y+60*	Move to starting point for group 2
N210 L2,0*	Call subprogram 2 for the group
N220 X+75 Y+10*	Move to starting point for group 3
N230 L2,0*	Call subprogram 2 for the group
N240 G98 L0*	End of subprogram 1
N250 G98 L2*	Beginning of subprogram 2: Group of holes
N260 G79*	Call cycle for 1st hole
N270 G91 X+20 M99*	Move to 2nd hole, call cycle
N280 Y+20 M99*	Move to 3rd hole, call cycle
N290 X-20 G90 M99*	Move to 4th hole, call cycle
N300 G98 L0*	End of subprogram 2
N310 %UP2 G71 *	

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.

Use Q parameters for e.g.:

- Coordinate values
- Feed rates
- Spindle speeds
- Cycle data

With Q parameters you can also:

- Program contours that are defined through mathematical functions
- Make execution of machining steps depend on certain logical conditions

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
	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)

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
	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 303
>
> 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)	261
TRIGO-NOMETRY	Trigonometric functions	264
JUMP	If/then conditions, jumps	266
DIVERSE FUNCTION	Other functions	270
FORMULA	Entering formulas directly	286
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 **d0: ASSIGN** assigns numerical values to Q parameters. This enables you to use variables in the NC program instead of fixed numerical values.

Example

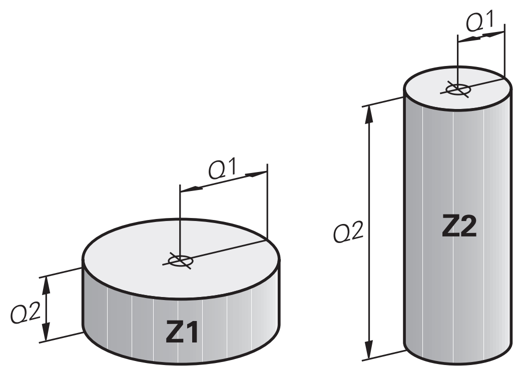
N150 D00 Q10 P01 +25*	Assign
...	Q10 is assigned the value 25
N250 G00 X +Q10*	Corresponds to G00 X +25

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
- To select the basic mathematical functions, press the **BASIC ARITHM...** soft key.
- > The control then displays the following soft keys:

Overview

Soft key	Function
D0 X = Y	**D00**: ASSIGN e. g., **D00 Q5 P01 +60 *** Directly assign value Reset Q parameter value
D1 X + Y	**D01**: ADDITION e. g., **D01 Q1 P01 -Q2 P02 -5 *** Calculate and assign the sum of two values
D2 X - Y	**D02**: SUBTRACTION e. g. **D02 Q1 P01 +10 P02 +5 *** Form and assign difference between two values
D3 X * Y	**D03**: MULTIPLICATION e. g. **D03 Q2 P01 +3 P02 +3 *** Form and assign the product of two values
D4 X / Y	**D04**: DIVISION e.g., **D04 Q4 P01 +8 P02 +Q2 *** Calculate and assign the quotient of two values **Not permitted:** Division by 0
D5 SQRT	**D05**: SQUARE ROOT e.g., **D05 Q50 P01 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

```
N16 D00 Q5 P01 +10*
N17 D03 Q12 P01 +Q5  P02 +7*
```

Q

- Select the Q parameter function: Press the **Q** key

BASIC ARITHM.

- To select the mathematical functions, press the **BASIC ARITHM.** soft key.

D0 X = Y

- To select the ASSIGN Q parameter function: Press the **D0 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.

- To select the mathematical functions, press the **BASIC ARITHM.** soft key.

D3 X * Y

- To select the MULTIPLICATION Q parameter function, press the **D3 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 D00: Q5 SET UNDEFINED*
17 D00: Q1 = Q5*
```

Q
- Select the Q parameter function: Press the **Q** key

BASIC ARITHM.
- To select the mathematical functions, press the **BASIC ARITHM.** soft key.

D0 X = Y
- To select the ASSIGN Q parameter function: Press the **D0 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?

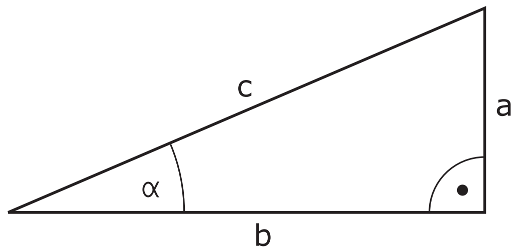
SET UNDEFINED
- Press **SET UNDEFINED**

> The **D00** function also supports transfer of the value **Undefined**. If you wish to transfer the undefined Q parameter without **D00**, the control shows the error message **Invalid value**.

9.4 Trigonometric functions

Definitions

Sine: $\sin \alpha = a / c$
Cosine: $\cos \alpha = b / c$
Tangent: $\tan\alpha = a / b = \sin \alpha / \cos \alpha$



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:
$\alpha = \arctan (a / b) = \arctan (\sin \alpha / \cos \alpha)$

Example:
a = 25 mm
b = 50 mm
$\alpha = \arctan (a / b) = \arctan 0.5 = 26.57°$
Furthermore:
$a^2 + b^2 = c^2$ (where $a^2 = a \times a$)
$c = \sqrt{(a^2 + b^2)}$

Programming trigonometric functions

Press the **TRIGONOMETRY** soft key to call the trigonometric functions. The control then displays the soft keys listed in the table below:

Soft key	Function
D6 SIN(X)	**D06**: SINUS e. g., **D06 Q20 P01 -Q5 *** Calculate and assign the sine of an angle in degrees (°)
D7 COS(X)	**D07**: COSINE e. g., **D07 Q21 P01 -Q5 *** Calculate and assign the cosine of an angle in degrees (°)
D8 X LEN Y	**D08**: ROOT SUM OF SQUARES e. g., **D08 Q10 P01 +5 P02 +4 *** Calculate and assign lengths from two values
D13 X ANG Y	**D13**: ANGLE e. g., **D13 Q20 P01 +10 P02 -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 Calculation of circles

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
D23 3 POINTS OF CIRCLE	FN 23: Determining the CIRCLE DATA from three points e. g., **D23 Q20 P01 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
D24 4 POINTS OF CIRCLE	FN 24: Determining the CIRCLE DATA from four points e. g., **D24 Q20 P01 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 **D23** and **D24** automatically overwrite the resulting parameter and the two following parameters.

9.6 If-then decisions with Q parameters

Application

The control can make logical if-then decisions by comparing a Q parameter with another Q parameter or with a numerical value. If the condition is fulfilled, the control continues the NC program at the label that is programmed after the condition.

Further information: "Labeling subprograms and program section repeats", Page 236

If it is not fulfilled, the control continues with the next NC block.

To call another NC program as a subprogram, enter a **%** program call after the block with the label.

Unconditional jumps

An unconditional jump is programmed by entering a conditional jump whose condition is always true. Example:

D09 P01 +10 P02 +10 P03 1 *

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**

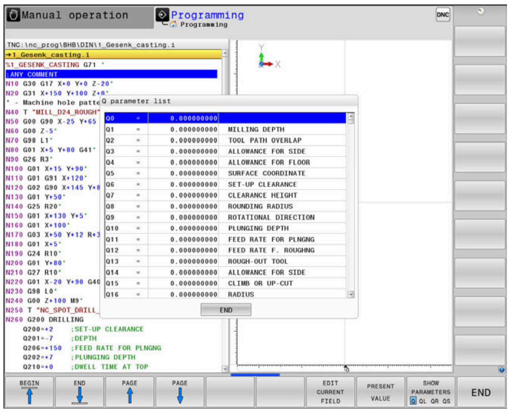
Press the **JUMP** soft key to call the if-then conditions. The control then displays the following soft keys:

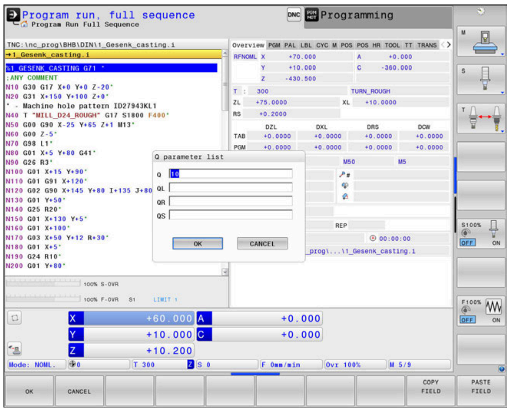
Soft key	Function
D9 IF X EQ Y GOTO	**D09**: IF EQUAL, JUMP e. g. **D09 P01 +Q1 P02 +Q3 P03 "UPCAN25" *** If both values or parameters are equal, jump to specified label
D9 IF X EQ Y GOTO IS UNDEFINED	**D09**: IF UNDEFINED, JUMP e. g., **D09 P01 +Q1 IS UNDEFINED P03 “UPCAN25“ *** If the specified parameter is undefined, then a jump is made to the specified label
D9 IF X EQ Y GOTO IS DEFINED	**D09**: IF DEFINED, JUMP e. g., **D09 P01 +Q1 IS DEFINED P03 “UPCAN25“ *** If the specified parameter is defined, then a jump is made to the specified label
D10 IF X NE Y GOTO	**D10**: IF UNEQUAL, JUMP e. g.**D10 P01 +10 P02 -Q5 P03 10 *** If both values or parameters are unequal, jump to specified label
D11 IF X GT Y GOTO	**D11**: IF GREATER, JUMP g. g.**D11 P01 +Q1 P02 +10 P03 QS5 *** If the first value or parameter is greater than the second value or parameter, jump to specified label
D12 IF X LT Y GOTO	**D12**: IF LESS, JUMP e. g. **D12 P01 +Q5 P02 +0 P03 "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

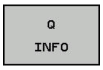
Procedure

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.
- If you would like to change the value, press the **EDIT CURRENT FIELD** soft key. Enter a 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

- Call the soft key row for screen layout

PROGRAM + STATUS

- Select the layout option for the additional status display
- > In the right half of the screen, the control shows the **Overview** status form.

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 control. 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

Press the **DIVERSE FUNCTION** soft key to call the additional functions. The control then displays the following soft keys:

Soft key	Function	Page
D14 ERROR=	**D14** Display error messages	271
D16 F-PRINT	**D16** Formatted output of texts or Q parameter values	275
D18 SYS-DATUM READ	**D18** Read system data	282
D19 PLC=	**D19** Transfer values to the PLC	282
D20 WAIT FOR	**D20** NC and PLC synchronization	283
D26 OPEN THE TABLE	**D26** Open a freely definable table	340
D27 WRITE TO TABLE	**D27** Write to a freely definable table	340
D28 READ TABLE	**D28** Read from a freely definable table	341
D29 PLC LIST=	**D29** Transfer up to eight values to the PLC	284
D37 EXPORT	**D37** Export local Q parameters or QS parameters into a calling NC program	285
D38 TRANSMIT	**D38** Send information from the NC program	285

D14: Displaying error messages

With the **D14** 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 **D14** 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.

```
N180 D14  P01 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

D16 – Formatted output of text and Q parameter values

Basics

With the function **D16**, 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 **D16** 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.

In machine parameters (no. 102202) and (no. 102203) you can define a default path for outputting log files.

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:

- Press the **PGM MGT** key

- 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
\n	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 D16 function. Example: "Measuring program: %S",CALL_PATH;
M_CLOSE	Closes the file to which you are writing with D16. 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 D16 output in an NC program

Within the **D16** you specify the output file that contains the texts to be output.

The control generates the output file:

- at the end of the program (**G71**),
- 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 D16.

Proceed as follows:

Q

- Press the **Q** key.

DIVERSE FUNCTION

- Press the **DIVERSE FUNCTION** soft key

D16 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 D16 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 **D16** function is located.

Program relative paths as an alternative to complete paths:

- Starting from the folder of the calling file one folder level down **D16 P01 MASKE\MASKE1.A/ PROT\PROT1.TXT**
- Starting from the folder of calling file one folder level up and in another folder **D16 P01 ..\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 **D16** 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 format of the output (e.g., TXT, .A, .XLS, .HTML).
- If you use **D16**, then no UTF-8 encoding is permitted for the file.
- Use **D18** 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: "D18 – Reading system data", Page 282

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 291

Enter Q parameters in the **D16** 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

```
N90 D16 P01 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 **D16** 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

```
N90 D16 P01 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

```
N90 D16 P01 TNC:\MASKE\MASKE1.A/SCLR:
```

Exporting messages

With the **D16** function you can also store log files externally.

To do so you must enter the target path in the **D16** function.

Example

```
N90 D16 P01 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 **D16** 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

```
N90 D16 P01 TNC:\MASKE\MASKE1.A/PRINTER:\DRUCK1
```

D18 – Reading system data

With the **D18** 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 **D18** are always output by the control in **metric** units regardless of the NC program's unit of measure.

Further information: "System data", Page 486

Example: Assign the value of the active scaling factor for the Z axis to Q25.

```
N55 D18 Q25 ID210 NR4 IDX3*
```

D19 – Transfer 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 **D19** function transfers up to two numerical values or Q parameters to the PLC.

D20 – 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 **D20** 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 **D20** block is fulfilled.

SYNC is used whenever you read, for example, system data via **D18** 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

```
N32 D20 SYNC
N33 D18 Q1 ID270 NR1 IDX1*
```

D29 – 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 **D29** function transfers up to eight numerical values or Q parameters to the PLC.

D37 - 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 **D37** function if you want to create your own cycles and integrate them in the control.

D38 – Send information from NC program

The function **D38** enables you to write texts and Q parameter values to the log from the NC program and send to a DNC application.

Further information: "D16 – Formatted output of text and Q parameter values", Page 275

Data transmission is through a standard TCP/IP computer network.

For more detailed information, consult the Remo Tools SDK manual.

Example

Document values from Q1 and Q23 in the log.

```
D38* /"Q parameter Q1: %f Q23: %f" P02 +Q1 P02 +Q23*
```

9.9 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 310

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
```

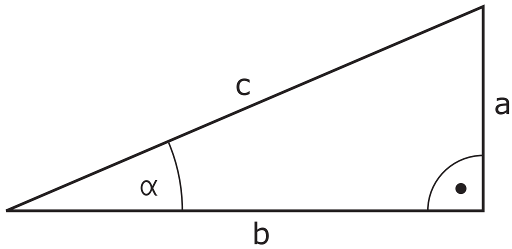
1. Calculation step 10 squared = 100
2. Calculation step 3 to the third power = 27
3. Calculation 100 – 27 = 73

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); then store in 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 tangentfunction

◁

(

- 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

```
N10 Q25 = ATAN (Q12/Q13)
```

9.10 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 **D16** 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 256

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	291
CFGREAD	Read out machine parameter	300
	Chain-linking string parameters	291
TOCHAR	Converting a numerical value to a string parameter	293
SUBSTR	Copy a substring from a string parameter	294
SYSSTR	Read system data	295

Soft key	Formula string functions	Page
TONUMB	Converting a string parameter to a numerical value	296
INSTR	Checking a string parameter	297
STRLEN	Finding the length of a string parameter	298
STRCOMP	Compare alphabetic priority	299

When you use the **STRING FORMULA** function, the result of the arithmetic operation is always a string. When you use the **FORMULA** function, the result of the 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

```
N30 DECLARE character 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
- Press the **STRING FORMULA** soft key

ENT
- 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

```
N37 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

```
N37 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)

```
N37 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 G39 PGM CALL**
	10	Path of the NC program selected with **%: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

```
N37 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

> 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.

```
N37 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

```
N37 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

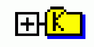
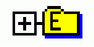
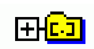
Example: QS12 and QS14 are compared for alphabetic priority

```
N37 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

N10 QS11 = "CH_NC"	Assign string parameter for key
N20 QS12 = "CfgGeoCycle"	Assign string parameter for entity
N30 QS13 = "pocketOverlap"	Assign string parameter for parameter name
N40 Q50 = CFGREAD(KEY_QS11 TAG_QS12 ATR_QS13)	Read out machine parameter

9.11 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 **G99** block)
- Delta value DR from the tool table
- Delta value DR from the **T** 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:

Tool axis	Parameter value
No tool axis defined	Q109 = –1
X axis	Q109 = 0
Y axis	Q109 = 1
Z axis	Q109 = 2
U axis	Q109 = 6
V axis	Q109 = 7
W axis	Q109 = 8

Spindle status: Q110

The value of the parameter Q110 depends on the M function last programmed for the spindle.

M function	Parameter value
No spindle status defined	Q110 = –1
M3: Spindle ON, clockwise	Q110 = 0
M4: Spindle ON, counterclockwise	Q110 = 1
M5 after M3	Q110 = 2
M5 after M4	Q110 = 3

Coolant on/off: Q111

M function	Parameter value
M8: Coolant ON	Q111 = 1
M9: Coolant OFF	Q111 = 0

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 **%**, the value of the parameter Q113 depends on the dimensional data of the NC program from which the other NC programs are called.

Dimensional data of the main program	Parameter value
Metric system (mm)	Q113 = 0
Imperial system (inch)	Q113 = 1

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 refer to the datum point that is active in the **Manual operation** mode.

The length of the stylus and the radius of the ball tip are not compensated in these coordinates.

Coordinate axis	Parameter value
X axis	Q115
Y axis	Q116
Z axis	Q117
4th axis Machine-dependent	Q118
5th axis Machine-dependent	Q119

Deviation between actual value and nominal value during automatic tool measurement with, for example, the TT 160

Deviation of actual from nominal value	Parameter value
Tool length	Q115
Tool radius	Q116

Tilting the working plane with spatial (workpiece) angles instead of spindle head angles: Coordinates for rotary axes calculated by the control.

Coordinates	Parameter value
A axis	Q120
B axis	Q121
C axis	Q122

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.

Status	Parameter value
No error	Q601 = 1
Error	Q601 = 2
No monitoring area defined or not enough reference images	Q601 = 3
Internal error (no signal, camera error, etc.)	Q601 = 10

9.12 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.

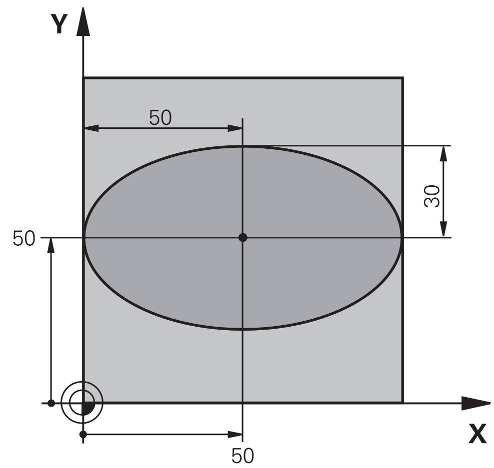
The control uses the **SGN** function to detect whether a number is positive or negative.

%ROUND G71 *	
N10 D00 Q1 P01 +34.789*	First number to be rounded
N20 D00 Q2 P01 +34.345*	Second number to be rounded
N30 D00 Q3 P01 -34.345*	Third number to be rounded
N40 ;	
N50 Q11 = INT (Q1 + 0.5 * SGN Q1)	Add the value 0.5 to Q1, then truncate the decimal places
N60 Q12 = INT (Q2 + 0.5 * SGN Q2)	Add the value 0.5 to Q2, then truncate the decimal places
N70 Q13 = INT (Q3 + 0.5 * SGN Q3)	Subtract the value 0.5 from Q3, then truncate the decimal places
N99999999 %ROUND G71 *	

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



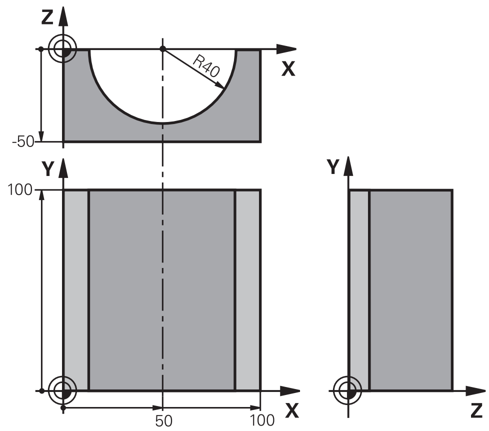
%ELLIPSE G71 *	
N10 D00 Q1 P01 +50*	Center in X axis
N20 D00 Q2 P01 +50*	Center in Y axis
N30 D00 Q3 P01 +50*	Semiaxis in X
N40 D00 Q4 P01 +30*	Semiaxis in Y
N50 D00 Q5 P01 +0*	Starting angle in the plane
N60 D00 Q6 P01 +360*	End angle in the plane
N70 D00 Q7 P01 +40*	Number of calculation steps
N80 D00 Q8 P01 +30*	Rotational position of the ellipse
N90 D00 Q9 P01 +5*	Milling depth
N100 D00 Q10 P01 +100*	Feed rate for plunging
N110 D00 Q11 P01 +350*	Feed rate for milling
N120 D00 Q12 P01 +2*	Set-up clearance for pre-positioning
N130 G30 G17 X+0 Y+0 Z-20*	Workpiece blank definition
N140 G31 G90 X+100 Y+100 Z+0*	
N150 T1 G17 S4000*	Tool call
N160 G00 G40 G90 Z+250*	Retract the tool
N170 L10.0*	Call machining operation
N180 G00 Z+250 M2*	Retract the tool, end program
N190 G98 L10*	Subprogram 10: Machining operation
N200 G54 X+Q1 Y+Q2*	Shift datum to center of ellipse
N210 G73 G90 H+Q8*	Account for rotational position in the plane
N220 Q35 = (Q6 - Q5) / Q7	Calculate angle increment
N230 D00 Q36 P01 +Q5*	Copy starting angle
N240 D00 Q37 P01 +0*	Set counter
N250 Q21 = Q3 * COS Q36	Calculate X coordinate for starting point
N260 Q22 = Q4 * SIN Q36	Calculate Y coordinate for starting point
N270 Q00 G40 X+Q21 Y+Q22 M3*	Move to starting point in the plane

N280 Z+Q12*	Pre-position in spindle axis to set-up clearance
N290 G01 Z-Q9 FQ10*	Move to working depth
N300 G98 L1*	
N310 Q36 = Q36 + Q35	Update the angle
N320 Q37 = Q37 + 1	Update the counter
N330 Q21 = Q3 * COS Q36	Calculate the current X coordinate
N340 Q22 = Q4 * SIN Q36	Calculate the current Y coordinate
N350 G01 X+Q21 Y+Q22 FQ11*	Move to next point
N360 D12 P01 +Q37 P02 +Q7 P03 1*	Unfinished? If not finished, return to LBL 1
N370 G73 G90 H+0*	Reset the rotation
N380 G54 X+0 Y+0*	Reset the datum shift
N390 G00 G40 Z+Q12*	Move to set-up clearance
N400 G98 L0*	End of subprogram
N99999999 %ELLIPSE G71 *	

Example: Concave cylinder machined with Ball-nose cutter

Program run

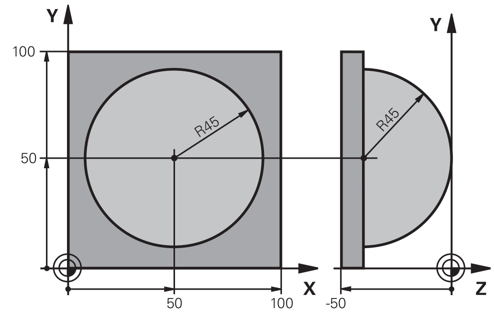
- This NC program functions only with a Ball-nose cutter. The tool length refers to the sphere center.
- The contour of the cylinder is approximated by many short line segments (defined in Q13). The more line segments you define, the smoother the curve becomes.
- The cylinder is milled in longitudinal 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



%CYLIN G71 *	
N10 D00 Q1 P01 +50*	Center in X axis
N20 D00 Q2 P01 +0*	Center in Y axis
N30 D00 Q3 P01 +0*	Center in Z axis
N40 D00 Q4 P01 +90*	Starting angle in space (Z/X plane)
N50 D00 Q5 P01 +270*	End angle in space (Z/X plane)
N60 D00 Q6 P01 +40*	Cylinder radius
N70 D00 Q7 P01 +100*	Length of the cylinder
N80 D00 Q8 P01 +0*	Rotational position in the X/Y plane
N90 D00 Q10 P01 +5*	Allowance for cylinder radius
N100 D00 Q11 P01 +250*	Feed rate for plunging
N110 D00 Q12 P01 +400*	Feed rate for milling
N120 D00 Q13 P01 +90*	Number of cuts
N130 G30 G17 X+0 Y+0 Z-50*	Workpiece blank definition
N140 G31 G90 X+100 Y+100 Z+0*	
N150 T1 G17 S4000*	Tool call
N160 G00 G40 G90 Z+250*	Retract the tool
N170 L10.0*	Call machining operation
N180 D00 Q10 P01 +0*	Reset allowance
N190 L10.0*	Call machining operation
N200 G00 G40 Z+250 M2*	Retract the tool, end program
N210 G98 L10*	Subprogram 10: Machining operation
N220 Q16 = Q6 - Q10 - Q108	Account for allowance and tool, based on the cylinder radius
N230 D00 Q20 P01 +1*	Set counter
N240 D00 q24 p01 +Q4*	Copy starting angle in space (Z/X plane)
N250 Q25 = (Q5 - Q4) / Q13	Calculate angle increment
N260 G54 X+Q1 Y+Q2 Z+Q3*	Shift datum to center of cylinder (X axis)

N270 G73 G90 H+Q8*	Account for rotational position in the plane
N280 G00 G40 X+0 Y+0*	Pre-position in the plane to the cylinder center
N290 G01 Z+5 F1000 M3*	Pre-position in the spindle axis
N300 G98 L1*	
N310 I+0 K+0*	Set pole in the Z/X plane
N320 G11 R+Q16 H+Q24 FQ11*	Move to starting position on cylinder, plunge-cutting obliquely into the material
N330 G01 G40 Y+Q7 FQ12*	Longitudinal cut in Y+ direction
N340 D01 Q20 P01 +Q20 P02 +1*	Update the counter
N350 D01 Q24 P01 +Q24 P02 +Q25*	Update solid angle
N360 D11 P01 +Q20 P02 +Q13 P03 99*	Finished? If finished, jump to end
N370 G11 R+Q16 H+Q24 FQ11*	Move on an approximated arc for the next longitudinal cut
N380 G01 G40 Y+0 FQ12*	Longitudinal cut in Y– direction
N390 D01 Q20 P01 +Q20 P02 +1*	Update the counter
N400 D01 Q24 P01 +Q24 P02 +Q25*	Update solid angle
N410 D12 P01 +Q20 P02 +Q13 P03 1*	Unfinished? If not finished, return to LBL 1
N420 G98 L99*	
N430 G73 G90 H+0*	Reset the rotation
N440 G54 X+0 Y+0 Z+0*	Reset the datum shift
N450 G98 L0*	End of subprogram
N99999999 %CYLIN G71 *	

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

%SPHERE G71 *	
N10 D00 Q1 P01 +50*	Center in X axis
N20 D00 Q2 P01 +50*	Center in Y axis
N30 D00 Q4 P01 +90*	Starting angle in space (Z/X plane)
N40 D00 Q5 P01 +0*	End angle in space (Z/X plane)
N50 D00 Q14 P01 +5*	Angle increment in space
N60 D00 Q6 P01 +45*	Sphere radius
N70 D00 Q8 P01 +0*	Starting angle of rotational position in the X/Y plane
N80 D00 Q9 p01 +360*	End angle of rotational position in the X/Y plane
N90 D00 Q18 P01 +10*	Angle increment in the X/Y plane for roughing
N100 D00 Q10 P01 +5*	Allowance in sphere radius for roughing
N110 D00 Q11 P01 +2*	Set-up clearance for pre-positioning in the spindle axis
N120 D00 Q12 P01 +350*	Feed rate for milling
N130 G30 G17 X+0 Y+0 Z-50*	Workpiece blank definition
N140 G31 G90 X+100 Y+100 Z+0*	
N150 T1 G17 S4000*	Tool call
N160 G00 G40 G90 Z+250*	Retract the tool
N170 L10.0*	Call machining operation
N180 D00 Q10 P01 +0*	Reset allowance
N190 D00 Q18 P01 +5*	Angle increment in the X/Y plane for finishing
N200 L10.0*	Call machining operation
N210 G00 G40 Z+250 M2*	Retract the tool, end program
N220 G98 L10*	Subprogram 10: Machining operation
N230 D01 Q23 P01 +Q11 P02 +Q6*	Calculate Z coordinate for pre-positioning
N240 D00 Q24 P01 +Q4*	Copy starting angle in space (Z/X plane)
N250 D01 Q26 P01 +Q6 P02 +Q108*	Compensate sphere radius for pre-positioning
N260 D00 Q28 P01 +Q8*	Copy rotational position in the plane
N270 D01 Q16 P01 +Q6 P02 -Q10*	Account for allowance in the sphere radius
N280 G54 X+Q1 Y+Q2 Z-Q16*	Shift datum to center of sphere
N290 G73 G90 H+Q8*	Account for starting angle of rotational position in the plane
N300 G98 L1*	Pre-position in the spindle axis

N310 I+0 J+0*	Set pole in the X/Y plane for pre-positioning
N320 G11 G40 R+Q26 H+Q8 FQ12*	Pre-position in the plane
N330 I+Q108 K+0*	Set pole in the Z/X plane, offset by the tool radius
N340 G01 Y+0 Z+0 FQ12*	Move to working depth
N350 G98 L2*	
N360 G11 G40 R+Q6 H+Q24 FQ12*	Move upward on an approximated arc
N370 D02 Q24 P01 +Q24 P02 +Q14*	Update solid angle
N380 D11 P01 +Q24 P02 +Q5 P03 2*	Inquire whether an arc is finished. If not finished, return to LBL 2
N390 G11 R+Q6 H+Q5 FQ12*	Move to the end angle in space
N400 G01 G40 Z+Q23 F1000*	Retract in the spindle axis
N410 G00 G40 X+Q26*	Pre-position for next arc
N420 D01 Q28 P01 +Q28 P02 +Q18*	Update rotational position in the plane
N430 D00 Q24 P01 +Q4*	Reset solid angle
N440 G73 G90 H+Q28*	Activate new rotational position
N450 D12 P01 +Q28 P02 +Q9 P03 1*	Unfinished? If not finished, return to LBL 1
N460 D09 P01 +Q28 P02 +Q9 P03 1*	
N470 G73 G90 H+0*	Reset the rotation
N480 G54 X+0 Y+0 Z+0*	Reset the datum shift
N490 G98 L0*	End of subprogram
N99999999 %SPHERE G71 *	

10

Special Functions

10.1 Overview of special functions

The control provides the following powerful special functions for a large number of applications:

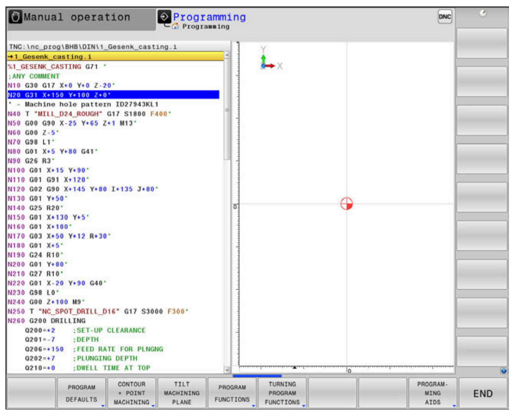
Function	Description
Dynamic Collision Monitoring with integrated fixture management (option 40)	Page 321
Adaptive Feed Control AFC (option 45)	Page 324
Active Chatter Control (option 145)	See the User's Manual for Setup, Testing and Running NC Programs
Working with text files	Page 333
Working with freely definable tables	Page 337

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



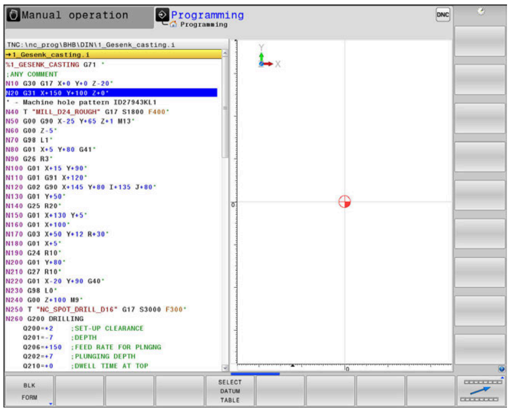
Soft key	Function	Description
PROGRAM DEFAULTS	Define program defaults	Page 319
CONTOUR + POINT MACHINING	Functions for contour and point machining	Page 319
TILT MACHINING PLANE	Define the **PLANE** function	Page 356
PROGRAM FUNCTIONS	Define different DIN/ISO functions	Page 320
TURNING PROGRAM FUNCTIONS	Define turning functions	Page 449
PROGRAM-MING AIDS	Programming aids	Page 181

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

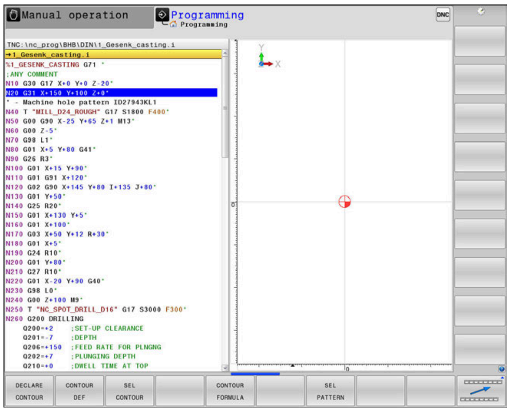


Soft key	Function	Description
BLK FORM	Define workpiece blank	Page 87
DATUM TABLE	Select datum table	See Cycle-Programming User's Manual
GLOBAL DEF	Define global cycle parameters	See Cycle-Programming User's Manual

Functions for contour and point machining menu

CONTOUR + POINT MACHINING

▶ Press the soft key for functions for contour and point machining

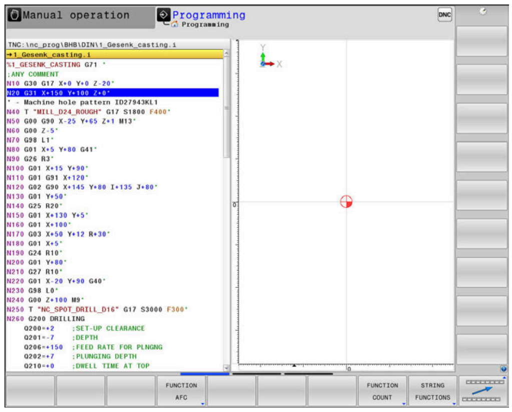


Soft key	Function	Description
DECLARE CONTOUR	Assign contour description	See Cycle-Programming User's Manual
CONTOUR DEF	Define a simple contour formula	See Cycle-Programming User's Manual
SEL CONTOUR	Select a contour definition	See Cycle-Programming User's Manual
CONTOUR FORMULA	Define a complex contour formula	See Cycle-Programming User's Manual
SEL PATTERN	Select the point file with machining positions	See Cycle-Programming User's Manual

Menu for defining different DIN/ISO functions

PROGRAM FUNCTIONS

- Press the **PROGRAM FUNCTIONS** soft key



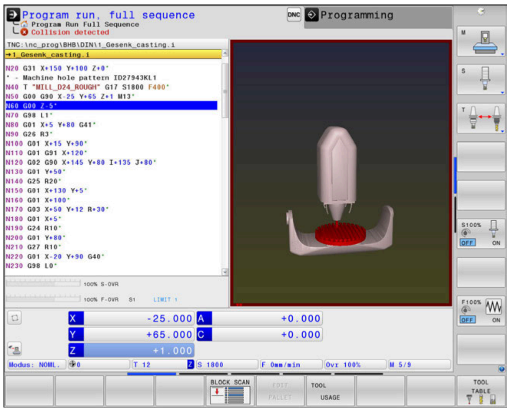
Soft key	Function	Description
FUNCTION AFC	Define Adaptive Feed Control	Page 324
FUNCTION COUNT	Define the counter	Page 331
STRING FUNCTIONS	Define string functions	Page 290
FUNCTION SPINDLE	Define pulsing spindle speed	Page 342
FUNCTION FEED	Define recurring dwell time	Page 344
FUNCTION DWELL	Define dwell time in seconds or revolutions	Page 346
FUNCTION DCM	Define Dynamic Collision Monitoring DCM	Page 321
DIN/ISO	Define DIN/ISO functions	Page 330
INSERT COMMENT	Add comments	Page 184
FUNCTION PROG PATH	Choose path interpretation	Page 393

10.2 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 **T** 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

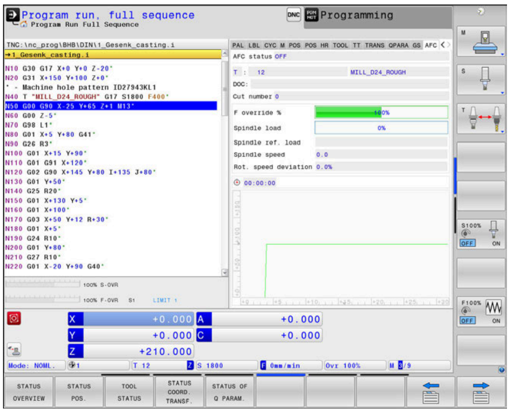
10.3 Adaptive Feed Control (AFC) (option 45)

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 only evaluates 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

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>.I.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>.I.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>.I.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 100

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

10.4 Defining DIN/ISO functions

Overview

> If an alphanumeric keyboard is connected via a USB port, you can also enter the ISO functions directly through the alphanumeric keyboard.

The control provides soft keys with the following functions for creating DIN/ISO programs:

Soft key	Function
DIN/ISO	Select ISO functions
F	Feed rate
G	Tool movements, cycles and program functions
I	X coordinate of the circle center or pole
J	Y coordinate of the circle center or pole
L	Label call for subprogram and program section repeat
M	Miscellaneous function
N	Block number
T	Tool call
H	Polar coordinate angle
K	Z coordinate of the circle center or pole
R	Polar coordinate radius
S	Spindle speed

10.5 Defining a counter

Application

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

The **FUNCTION COUNT** function allows you to control a simple counter from within the NC program. For example, this function allows you to count the number of manufactured workpieces.

Proceed as follows for the definition:

SPEC FCT

- Show the soft key row with special functions

PROGRAM FUNCTIONS

- Press the **PROGRAM FUNCTIONS** soft key

FUNCTION COUNT

- Press the **FUNCTION COUNT** soft key

NOTICE

Caution: Data may be lost!

Only one counter can be managed by the control. If you execute an NC program that resets the counter, any counter progress of another NC program will be deleted.

- Please check prior to machining whether a counter is active.
- If necessary, note down the counter value and enter it again via the MOD menu after execution.

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.

Define 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

N50 FUNCTION COUNT RESET*	Reset the counter value
N60 FUNCTION COUNT TARGET10*	Enter the target number of parts to be machined
N70 G98 L11*	Enter the jump label
N80 G ...	Machining
N510 FUNCTION COUNT INC*	Increment the counter value
N520 FUNCTION COUNT REPEAT LBL 11*	Repeat the machining operations if more parts are to be machined.
N530 M30*	
N540 %COUNT G71*	

10.6 Creating text files

Application

You can use the control's text editor to write and edit texts. Typical applications:

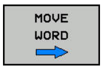
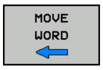
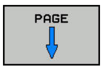
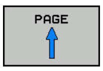
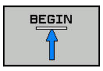
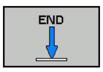
- Recording test results
- Documenting working procedures
- Creating formula collections

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 one after the other
- Select a file and open it with the **SELECT** soft key or **ENT** key, or create 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 cursor one word to the right
	Move cursor one word to the left
	Go to next screen page
	Go to previous screen page
	Cursor at beginning of file
	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 deleted 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:

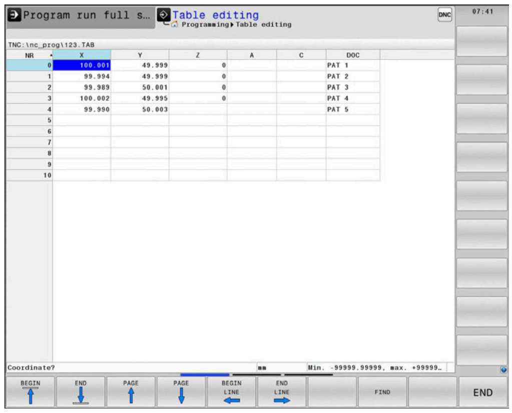
- Move the cursor to the desired word.
- To select the search function, press the **FIND** soft key.
- Press the **FIND CURRENT WORD** soft key.
- To find a word: press the **FIND** soft key.
- Exit the search function: Press the **END** soft key

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
- To find text: press the **FIND** soft key.
- Exit the search function: Press the **END** soft key

10.7 Freely definable tables

Fundamentals



In freely definable tables you can save and read any information from the NC program. The Q parameter functions **D26** to **D28** 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 inputting 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 338

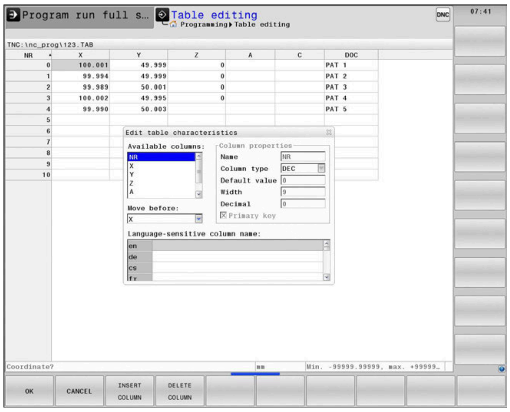
> Refer to your machine manual.
>
> Machine tool builders may define their own table templates and save them in the control. When you create a new table, the control opens a pop-up window listing all available table templates.

> 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:

- 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 characters)
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:

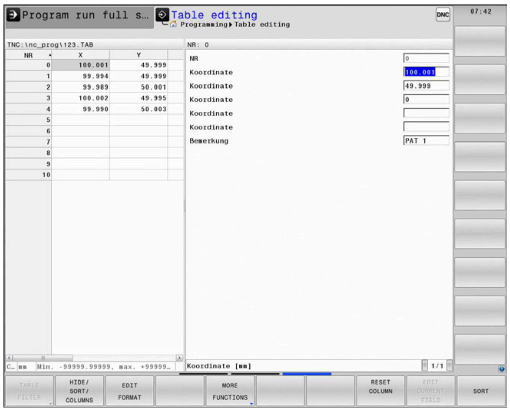
OK

- 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

D26 – Open a freely definable table

With the function **D26: TABOPEN** you open a freely definable table to be written to with **D27** or to be read from with **D28**.

> Only one table can be opened in an NC program at any one time. A new NC block with **D26** 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.

```
N56 D26  TNC:\DIR1\TAB1.TAB
```

D27 – Write to a freely definable table

With the **D27** function you write to the table that you previously opened with **D26**.

You can define multiple column names in a **D27** 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 **D27** function by default writes values to the currently open table, even in the **Test Run** operating mode. The **D18 ID992 NR16** function allows you to retrieve the operating mode in which the NC program is running. If the function **D27** may only be run in the operating modes **Program run, single block** and **Program run, full sequence**, then you can use a jump instruction to skip the corresponding program section.
>
> **Further information:** "If-then decisions with Q parameters", Page 266
>
> 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**.

```
N50 Q5 = 3,75
N60 Q6 = -5
N70 Q7 = 7,5
N80 D27  P01 5/“RADIUS,TIEFE,D“ = Q5
```

D28 – Read from a freely definable table

With the **D28** function you read from the table previously opened with **D26**.

You can define, i.e. read, multiple column names in a **D28** block. The column names must be written between quotation marks and separated by a comma. In the **D28** block you can define the Q parameter number in which the control is to write the value that is first read.

> **i** 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 Q parameter **Q10** (second value in **Q11**, third value in **Q12**).

From the same row, save the column **DOC** in **QS1**.

```
N50 D28  Q10 = 6/“X,Y,D“*
N60 D28  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. Existing data is not automatically backed up by the control before running the format change process, This permanently changes the files so that they 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

> **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 inputting data or reading it out.

10.8 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

```
N30 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

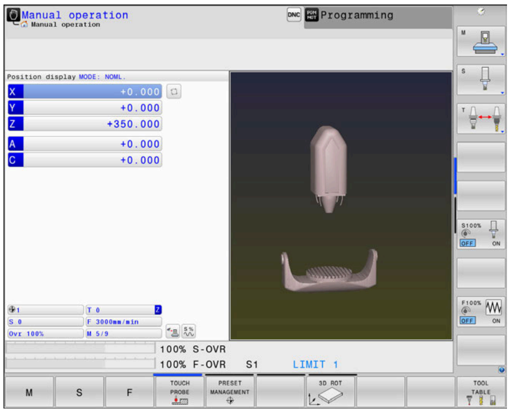
FUNCTION SPINDLE
- Press the **FUNCTION SPINDLE** soft key

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 symbol indicates the condition of the pulsing shaft speed:



Icon	Function
S %	Pulsing spindle speed active

Resetting the pulsing spindle speed

Example

```
N40 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.

10.9 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 while the spindle continues to turn. Due to this behavior, workpieces need to be scrapped if threads are cut. In addition, there is a danger of tool breakage during execution!

- Deactivate the **FUNCTION FEED DWELL** function before cutting threads

Procedure

Example

```
N30 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

```
N40 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.

10.10 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

```
N30 FUNCTION DWELL TIME10*
```

Example

```
N40 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

10.11 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 232

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 **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

```
N40 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

```
N40 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

```
N40 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	353
M116	Feed rate of rotary axes	384
PLANE/M128	Inclined-tool machining	383
M126	Shortest-path traverse of rotary axes	385
M94	Reduce display value of rotary axes	386
M128	Define the behavior of the control when positioning the rotary axes	387
M138	Selection of tilted axes	390
M144	Calculate machine kinematics	391

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: "Specifying the positioning behavior of the PLANE function", Page 372

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 **28 MIRROR IMAGE** may have different effects in conjunction with the **Tilt working plane** function. The effect mainly depends on the programming sequence, the mirrored axes and the tilting function used. There is a danger 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 **Program run, single block** operating mode

Examples

1 Cycle **28 MIRROR IMAGE** 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**

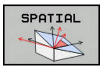
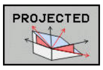
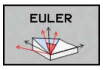
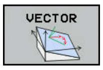
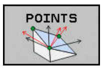
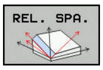
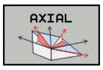
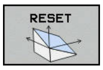
2 Cycle **28 MIRROR IMAGE** programmed before the tilting function with a rotary axis:
  - The mirrored rotary axis has no effect on the tilt specified in the **PLANE** function used, because only the movement of the rotary axis is mirrored

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.

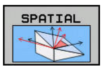
Overview

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:

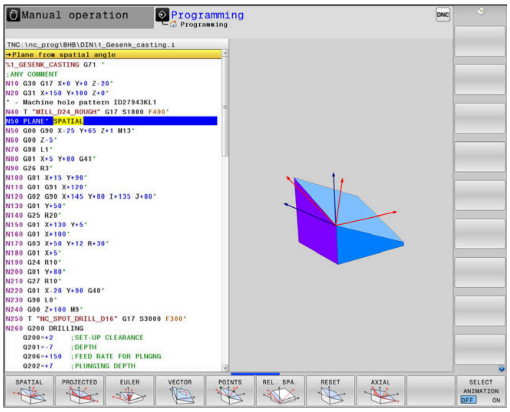
Soft key	Function	Required parameters	Page
	**SPATIAL**	Three spatial angles: **SPA**, **SPB**, and **SPC**	358
	**PROJECTED**	Two projection angles: **PROPR** and **PROMIN** and a rotation angle **ROT**	360
	**EULER**	Three Euler angles: precession (**EULPR**), nutation (**EULNU**) and rotation (**EULROT**),	362
	**VECTOR**	Normal vector for defining the plane and base vector for defining the direction of the tilted X axis	364
	**POINTS**	Coordinates of any three points in the plane to be tilted	367
	**RELATIVE**	Single, incrementally effective spatial angle	369
	**AXIAL**	Up to three absolute or incremental axis angles **A,B,C**	370
	**RESET**	Reset the PLANE function	357

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

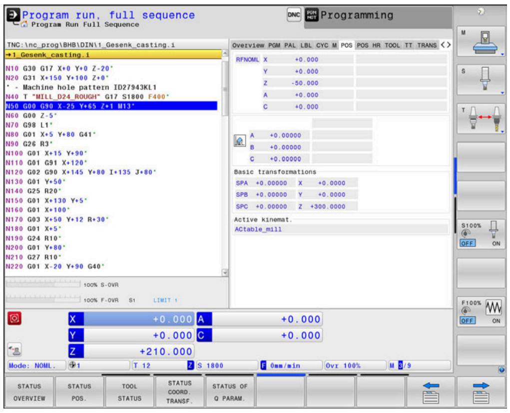
Selecting functions

- 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.

In the Distance-To-Go display (**ACTDST** and **REFDST**) the control shows, during tilting (**MOVE** or **TURN** mode) in the rotary axis, the distance to go to the calculated final position of the rotary axis.

Resetting PLANE function

Example

```
N10 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 the home position (**MOVE** or **TURN**) or not (**STAY**)
  Further information: "Automatic positioning: MOVE/TURN/STAY (input is mandatory)", Page 373

END

- Press the **END** key.

> The **PLANE RESET** function resets the active tilt and the angles (**PLANE** function or Cycle **G80**) (angle = 0 and function inactive). It does not need to be defined more than once.
>
> Deactivate tilting in the **Manual operation** mode in the 3D ROT menu.
>
> **Further information:** User's Manual for Setup, Testing and Running NC Programs

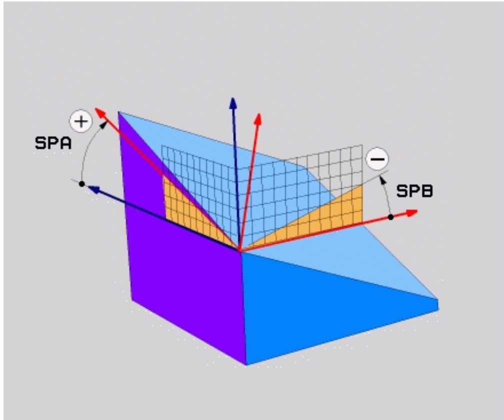
Defining the working plane with the spatial angle: PLANE SPATIAL

Application

Spatial angles define a working plane through up to three rotations in the non-tilted workpiece coordinate system (**tilting sequence A-B-C**).

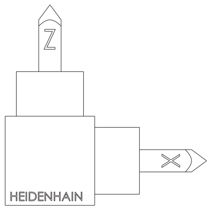
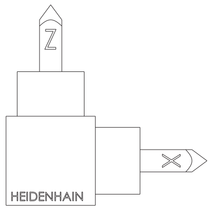
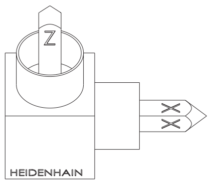
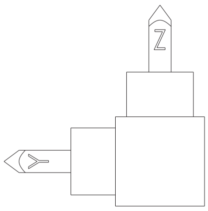
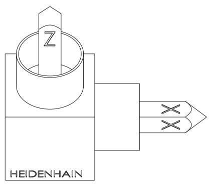
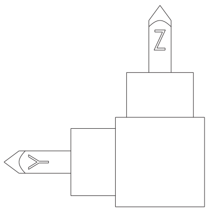
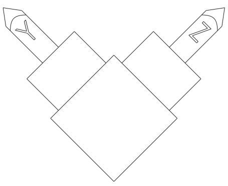
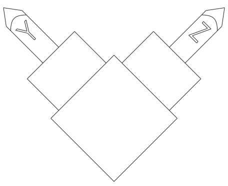
Most users assume three successive rotations in the reversed order (**tilting sequence C-B-A**).

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

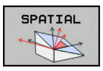
Programming notes:

- 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 **G80** 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 **G80** and in the **PLANE SPATIAL** function.
- You can select the desired positioning behavior.
  Further information: "Specifying the positioning behavior of the PLANE function", Page 372

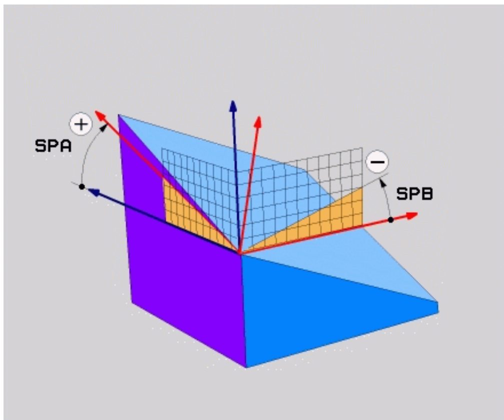
Input parameters

Example

```
N50 PLANE  SPATIAL SPA+27  SPB+0 SPC+45  .....*
```

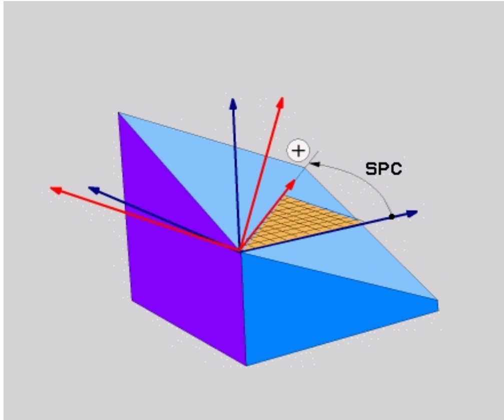


- **Spatial angle A?**: Rotational angle **SPA** about the (non-tilted) X axis. Input range from -359.9999 to +359.9999
- **Spatial angle B?**: Rotational angle **SPB** about the (non-tilted) Y axis. Input range from -359.9999 to +359.9999
- **Spatial angle C?**: Rotational angle **SPC** about the (non-tilted) Z axis. Input range from -359.9999 to +359.9999
- Continue with the positioning properties
  Further information: "Specifying the positioning behavior of the PLANE function", Page 372



Abbreviations used

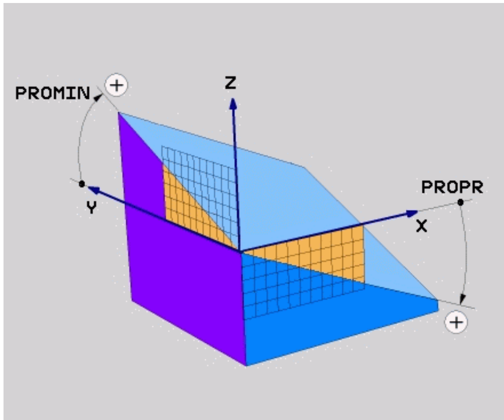
Abbreviation	Meaning
SPATIAL	In space
SPA	**Sp**atial **A**: Rotation about the (non-tilted) X axis
SPB	**Sp**atial **B**: Rotation about the (non-tilted) Y axis
SPC	**Sp**atial **C**: Rotation about the (non-tilted) Z axis



Defining the working plane with the projection angle: PLANE PROJECTED

Application

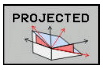
Projection angles define a working plane by specifying two angles that you can communicate by projection of the 1st coordinate plane (Z/X on tool axis Z) and 2nd coordinate plane (Y/Z on tool axis Z) to the working levels to be defined.



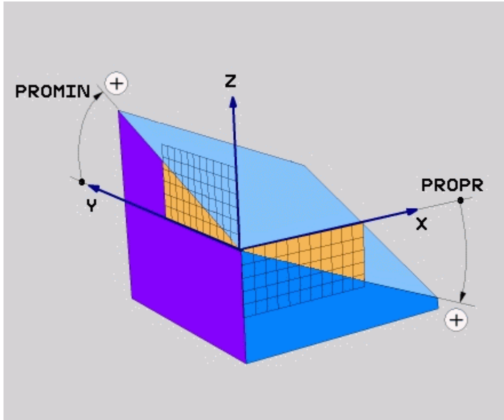
Programming notes:

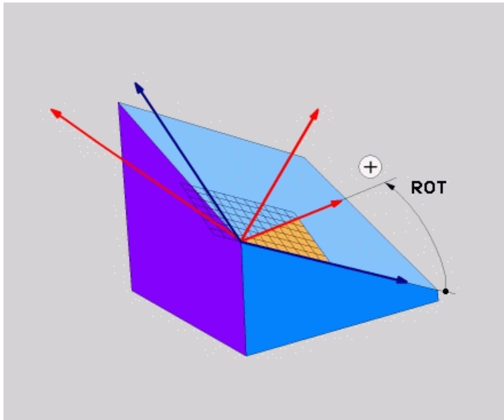
- The projection angles correspond to the angle projections on the planes of a rectangular coordinate system. The angles at the outer faces of the workpiece only are identical to the projection angles if the workpiece is rectangular. Thus, with workpieces that are not rectangular, the angle specifications from the engineering drawing often differ from the actual projection angles.
- You can select the desired positioning behavior. **Further information:** "Specifying the positioning behavior of the PLANE function", Page 372

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: "Specifying the positioning behavior of the PLANE function", Page 372





Example

```
N50 PLANE  PROJECTED PROPR+24  PROMIN+24 ROT+30  .....*
```

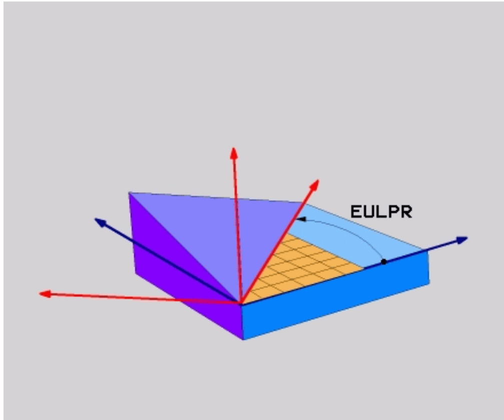
Abbreviations used:

PROJECTED	Projected
PROPR	Principal plane
PROMIN	Minor plane
ROT	Rotation

Defining the working plane with the Euler angle: PLANE EULER

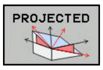
Application

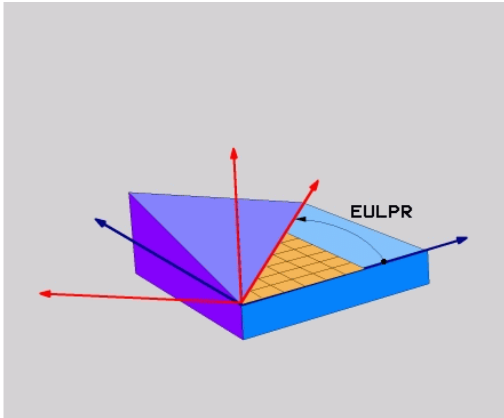
Euler angles define a machining plane through up to three **rotations about the respectively tilted coordinate system**. The Swiss mathematician Leonhard Euler defined these angles.



> You can select the desired positioning behavior.
> **Further information:** "Specifying the positioning behavior of the PLANE function", Page 372

Input parameters





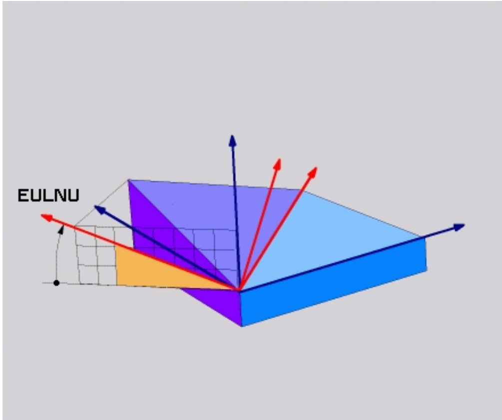
- **Rot. angle Main coordinate plane?:** Rotary angle **EULPR** around the Z axis. Please note:
  - Input range: -180.0000° to 180.0000°
  - The 0° axis is the X axis
- **Tilting angle tool axis?**: Tilting angle **EULNUT** of the coordinate system around the X axis shifted by the precession angle. Please 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: "Specifying the positioning behavior of the PLANE function", Page 372

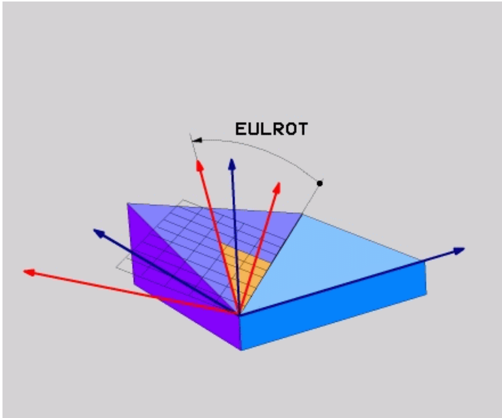
Example

```
N50 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



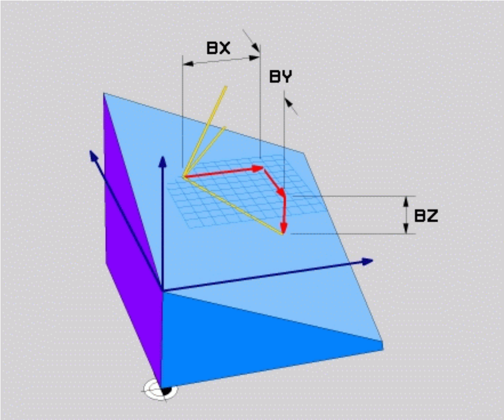


Defining the working plane with two vectors: PLANE VECTOR

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:** "Specifying the positioning behavior of the PLANE function", Page 372

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.

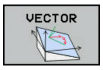
Default correction behavior of the control if the base vector is not perpendicular:

- The base vector is projected along the normal vector onto the working plane (defined by the normal vector).

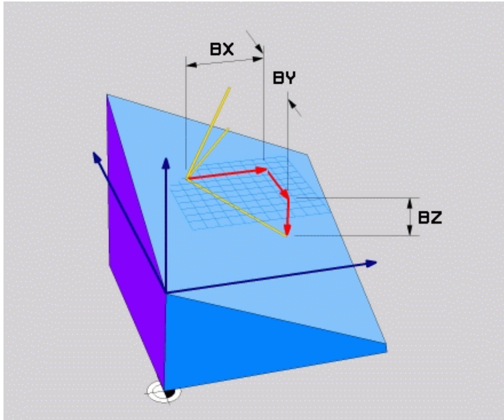
Correction behavior of the control if the base vector is not perpendicular and too short, parallel or antiparallel to the normal vector:

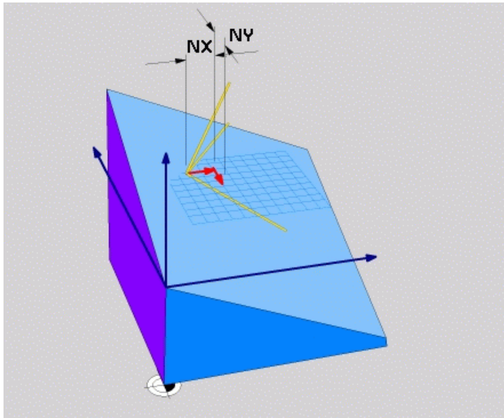
- If the normal vector has no X component, the base vector corresponds to the original X axis
- If the normal vector has no Y component, the base vector corresponds to the original Y axis

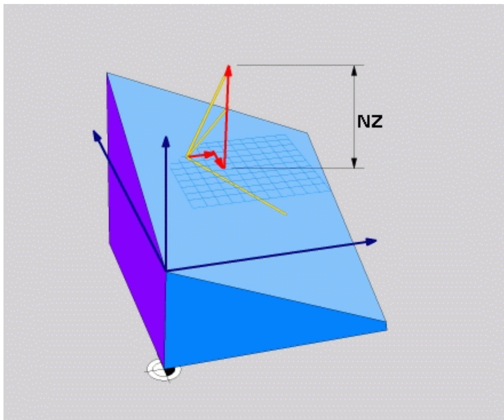
Input parameters



- **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: "Specifying the positioning behavior of the PLANE function", Page 372







Example

```
N50 PLANE  VECTOR BX0.8  BY-0.4 BZ-0.42  NX0.2 NY0.2 NT0.92 ..*
```

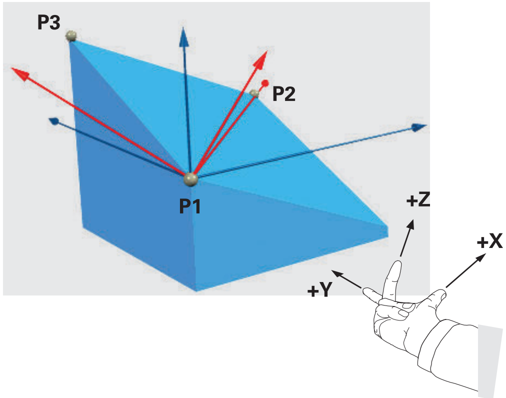
Abbreviations used

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

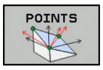
A working plane can be uniquely defined by entering **any three points P1 to P3 in this plane**. This possibility is realized in the **PLANE POINTS** function.



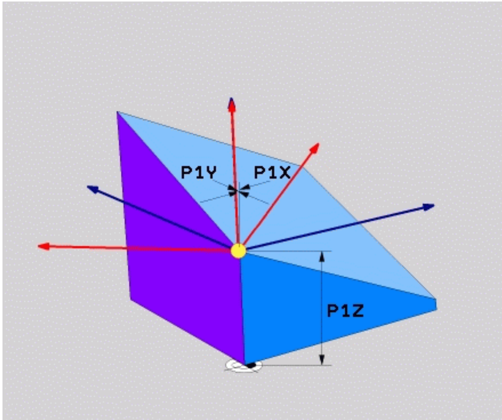
Programming notes:

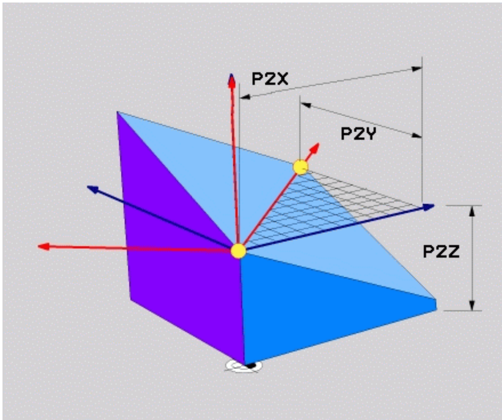
- The three points define the slope and orientation of the plane. The position of the active datum is not changed through **PLANE POINTS**.
- Point 1 and Point 2 determine the orientation of the tilted main axis X (for tool axis Z).
- Point 3 defines the slope of the tilted working plane. In the defined 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:** "Specifying the positioning behavior of the PLANE function", Page 372

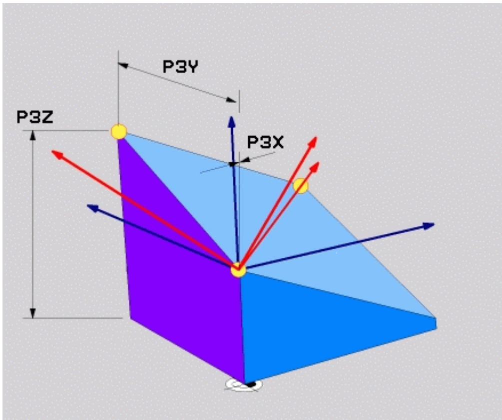
Input parameters



- **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: "Specifying the positioning behavior of the PLANE function", Page 372







Example

```
N50 PLANE  POINTS P1X+0  P1Y+0 P1Z+20  P2X+30 P2Y+31  P2Z+20
    P3X+0 P3Y+41  P3Z+32.5 .....*
```

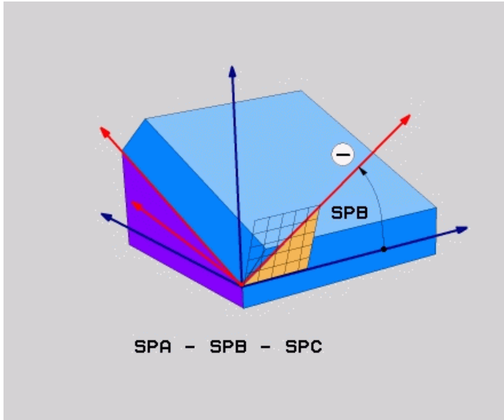
Abbreviations used

Abbreviation	Meaning
POINTS	Points

Defining the working plane via a single incremental spatial angle: PLANE RELATIV

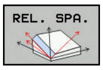
Application

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.

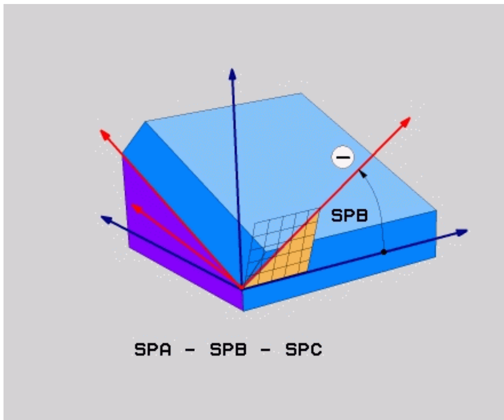


> Programming notes:
>
> - The defined angle is always in effect in respect to the active working plane, regardless of the tilting function you used before.
> - You can program any number of **PLANE RELATIV** functions in a row..
> - If you want to return the working plane to the orientation that was 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:** "Specifying the positioning behavior of the PLANE function", Page 372

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:** "Specifying the positioning behavior of the PLANE function", Page 372



Example

```
N50 PLANE RELATIV SPB-45 .....*
```

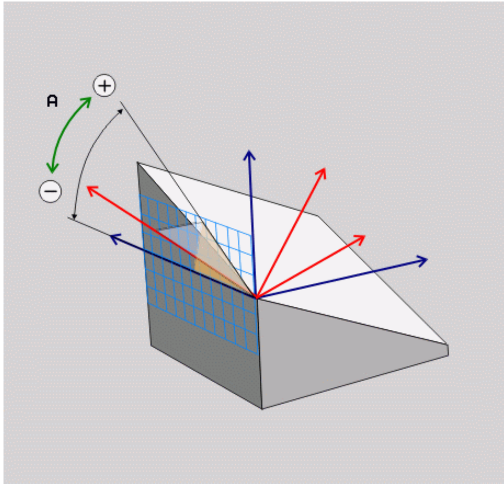
Abbreviations used

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**.

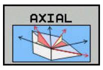
Programming notes:

- 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.

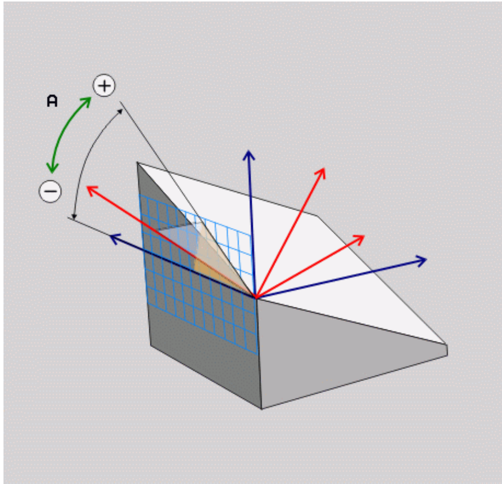
Input parameters

Example

```
N50 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:** "Specifying the positioning behavior of the PLANE function", Page 372



Abbreviations used

Abbreviation	Meaning
AXIAL	In the axial direction

Specifying 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 **28 MIRROR IMAGE** may have different effects in conjunction with the **Tilt working plane** function. The effect mainly depends on the programming sequence, the mirrored axes and the tilting function used. There is a danger 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 **Program run, single block** operating mode

Examples

1. Cycle **28 MIRROR IMAGE** 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. Cycle **28 MIRROR IMAGE** 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

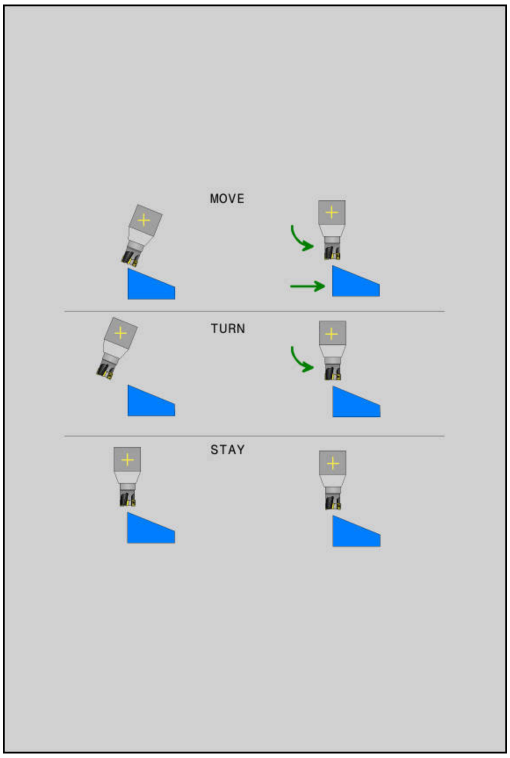
Automatic positioning: MOVE/TURN/STAY (input is mandatory)

After you have entered all parameters for the plane definition, you must specify how the rotary axes will be positioned to the calculated axis values:

- The PLANE function is to automatically position the rotary axes to the calculated position values. The position of the tool relative to the workpiece is to remain the same.
- > The control carries out a compensation movement in the linear axes.

- The PLANE function is to automatically position the rotary axes to the calculated position values, but only the rotary axes are positioned.
- > The control does **not** carry out a compensation movement for the linear axes.

- You will position the rotary axes later in a separate positioning block

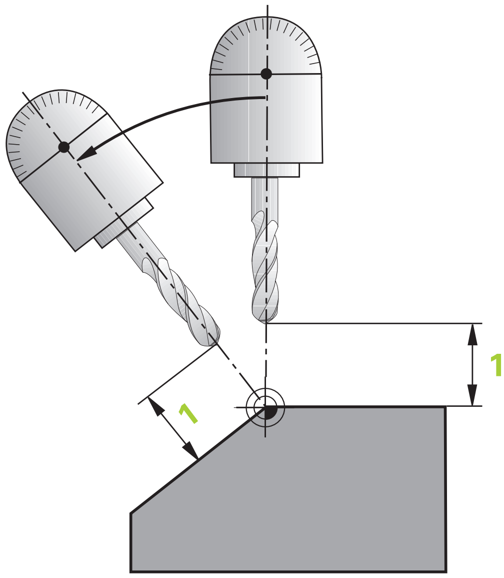


If you have selected the **MOVE** option (**PLANE** function is to position the axes automatically), the following two parameters must still be defined: **Dist. tool tip - center of rot.** and **Feed rate? F=**.

If you have selected the **TURN** option (**PLANE** function is to position the axes automatically without any compensating movement), the following parameter must still be defined: **Feed rate? F=**.

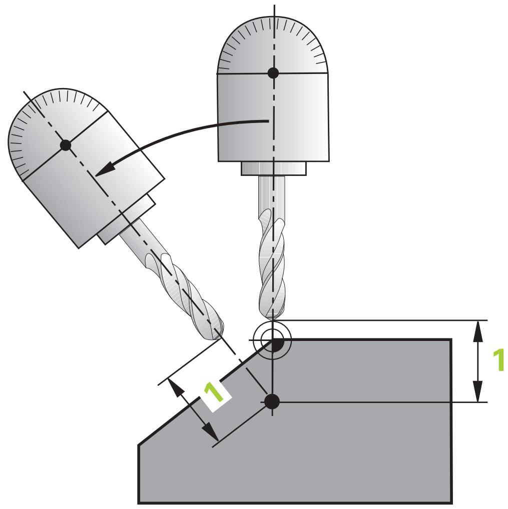
As an alternative to defining a feed rate **F** directly by entering a numerical value, you can also position the axes with **FMAX** (rapid traverse) or **FAUTO** (feed rate from the **T** block).

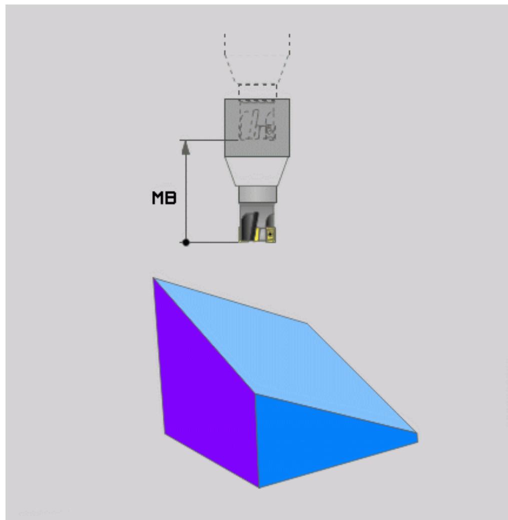
> If you use **PLANE** together with **STAY,** you have to position the rotary axes in a separate block after the **PLANE** function.



- **Dist. tool tip - center of rot.** (incremental): The **DIST** parameter shifts the center of rotation of the movement relative to the current position of the tool tip.
  - If the tool is already at the given distance to the workpiece before positioning, then it will be at the same relative position after positioning (see center figure at the right, **1** = DIST)
  - If the tool is not at the given distance to the workpiece before positioning, then it will be offset relatively from the original position after positioning (see lower figure at the right, **1** = DIST)
- > The control tilts the tool (or table) relative to the tool tip.

- **Feed rate? F=**: Contour speed to be used by the tool for positioning
- **Retraction length in the tool axis?**: The retraction path **MB** is effective incrementally from the current tool position in the active tool axis direction that the control approaches **before tilting**. **MB MAX** positions the tool just before the software limit switch.





Positioning the rotary axes in a separate NC block

To position the rotary axes 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 to position can lead to a risk of collision during the tilting movement!

- Program a safe position of the tool before the tilting movement.
- Carefully test the NC program or program section in **Program run, single block** operating mode

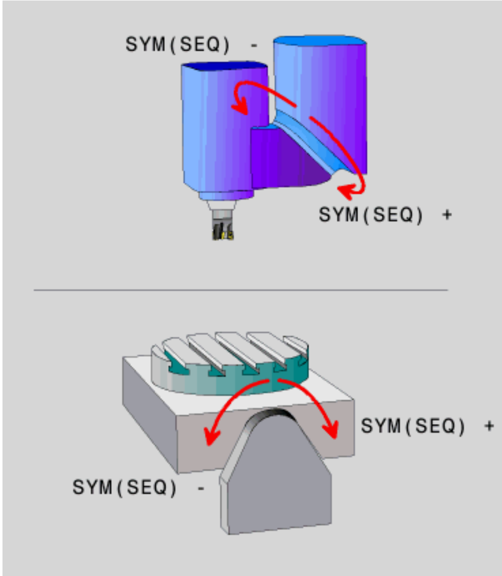
- Select any **PLANE** function, and define automatic tilting to position with the **STAY** option. During program execution, the control calculates the position values of the rotary axes present on the machine, and stores them in the system parameters Q120 (A axis), Q121 (B axis) and Q122 (C axis)
- 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

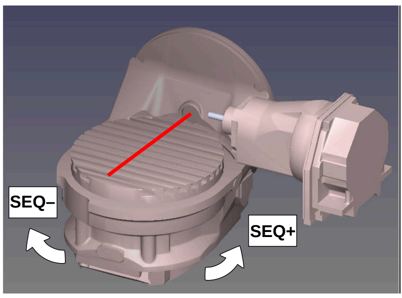
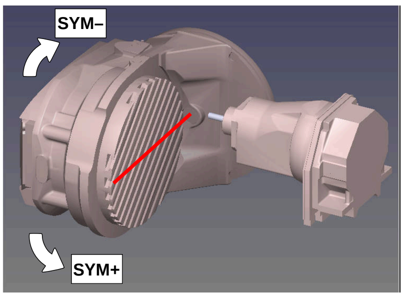
...	
N10 G00 Z+250 G40*	Position at clearance height
N20 PLANE SPATIAL SPA+0 SPB+45 SPC+0 STAY*	Define and activate the PLANE function
N30 G01 A+Q120 C+Q122 F2000*	Position the rotary axis with the values calculated by the control.
...	Define machining in the tilted working plane

Selection of alternate tilting possibilities: SYM (SEQ) +/– (entry optional)

The orientation you define for the working plane is used by the control to calculate the appropriate position of the rotary axes on your machine. In general, there are always two possible solutions.



> The control offers two variants (**SYM** and **SEQ**) for the selection of one of the possible solutions. You use soft keys to choose the variants. **SYM** is the standard variant.
>
> **SEQ** assumes that the master axis is in its home position (0°). The master axis is the first rotary axis from the tool, or the last rotary axis from the table (depending on the machine configuration). If both possible solutions are in the positive or negative range, 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 possibility) before tilting the working plane, or work with **SYM**.
>
> As opposed to **SEQ**, **SYM** uses the symmetry point of the master axis as reference. Every master axis has two symmetry positions, which are 180° apart from each other (sometimes only one symmetry position is within 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
> - Form 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 a possible solution with reference to the symmetry point of the master axis:

- **SYM+** positions the master axis in the positive half-space seen from the symmetry point.
- **SYM-** positions the master axis in the negative half-space seen from the symmetry point.

With the **SEQ** function you select a possible solution with reference to the home position of the master axis:

- **SEQ+** positions the master axis in the positive tilting range seen from the home position.
- **SEQ-** positions the master axis in the negative tilting range seen from the home position.

If the solution you chose with **SYM** (**SEQ**) is not within the machine's range of traverse, the control displays the **Entered angle not permitted** error message.

> If the **PLANE AXIAL** function is used, the **SYM** (**SEQ**) function has no effect.

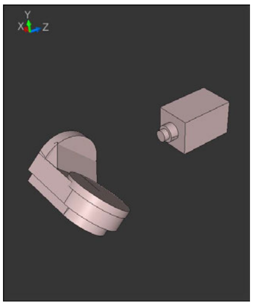
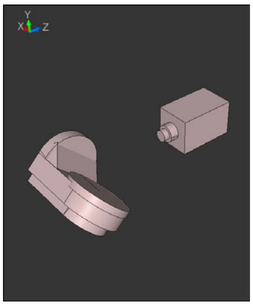
If you do not define **SYM** (**SEQ**), the control determines the solution as follows:

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**

Example for a machine with a rotary table C and a tilting table A. Programmed function: PLANE SPATIAL SPA+0 SPB+45 SPC+0

Limit switch	Starting 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

Example for a machine with a rotary table B and a tilting table A (limit switches for 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 depends on the kinematics. If you change the kinematics (such as switching the head), then the position of the symmetry point also changes.

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.

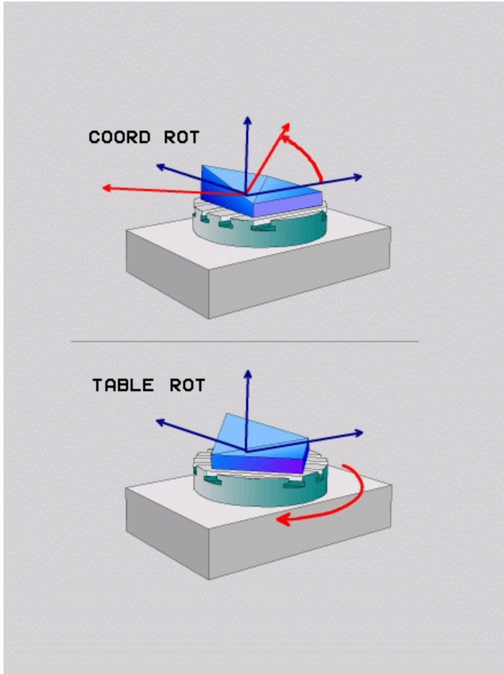
Selecting the type of transformation (input optional)

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.

Any rotary axis becomes a free rotary axis with the following constellation:

- 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.

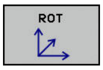
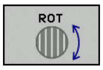


Programming notes:

- If no free rotary axis is created in a tilting situation, 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 does not matter if the free rotary axis is a table or 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 with Cycle 10 **ROTATION**

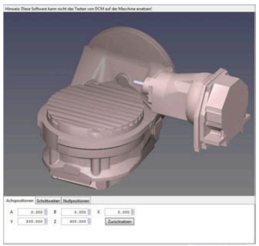
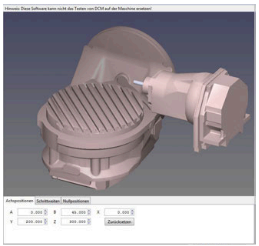
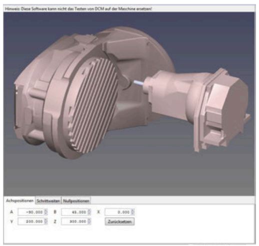
Soft key	Effect
	**COORD ROT**: > The control positions the free rotary axis to 0 > The control aligns the working plane coordinate system according to the programmed spatial angle
	**TABLE ROT** with: ■ SPA **and** SPB **equal to** 0 ■ SPC **equal or unequal to** 0 > The control aligns the free rotary axis according to the programmed spatial angle > The control aligns the working plane coordinate system according to 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 before tilting the working plane is maintained. > Since the workpiece was not positioned, the control aligns the working plane coordinate system according to the programmed spatial angle.

If no transformation type was specified, the control uses the **COORD ROT** transformation type for the **PLANE** functions

Example

The example below shows the effect of the **TABLE ROT** transformation type in conjunction with a free rotary axis.

...	
N60 G00 B+45 R0*	Pre-position rotary axis
N70 PLANE SPATIAL SPA-90 SPB+20 SPC+0 TURN F5000 TABLE ROT*	Tilt 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 before tilting the working plane is maintained
- Since the workpiece was not positioned, the control aligns the working plane coordinate system according to 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

```
N10  T 5 G17  S4500*
N20  PLANE SPATIAL SPA+0 SPB-90 SPC+0 STAY*
```

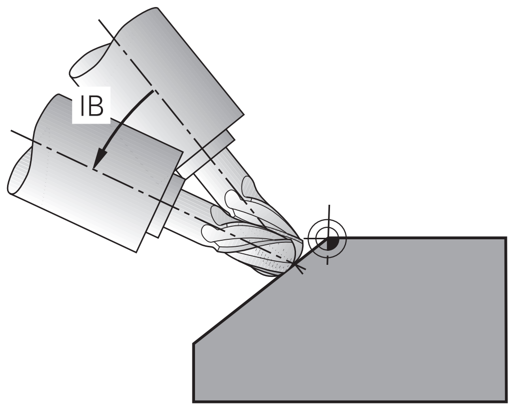
> The tilt angle must be precisely adapted to the tool angle, otherwise the control will generate an error message.

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 in a tilted machining plane only works with spherical cutters.

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

...	
N12 G00 G40 Z+50*	Position at clearance height
N13 PLANE SPATIAL SPA+0 SPB-45 SPC+0 MOVE DIST50 F900*	Define and activate the PLANE function
N14 M128*	Activate M128
N15 G01 G91 F1000 B-17*	Set the incline angle
...	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 description of kinematics.

Programming notes:

- The **M116** function can be used with table axes and head axes.
- The **M116** function is also effective if the **Tilt working plane** function is active.
- It is not possible to combine the **M128** or **TCPM** functions with **M116**. If you want to activate **M116** for an axis while the **M128** or **TCPM** function is active, you must deactivate the compensating movement for this axis indirectly using **M138**. This is done indirectly because with **M138**, you specify the axis for which the **M128** or **TCPM** function is effective. Thus, **M116** automatically affects the very axis that was not selected with **M138**.
  Further information: "Selecting tilting axes: M138", Page 390
- Without the **M128** or **TCPM** function, **M116** can be effective 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.

Shortest-path traverse of rotary axes: M126

Standard behavior

Refer to your machine manual.
The positioning behavior of rotary axes is machine-dependent.

The default behavior of the control while positioning rotary axes whose display has been reduced to values less than 360° is dependent on the **shortestDistance** machine parameter (no. 300401). This machine parameter defines whether the control should consider the difference between nominal and actual positions, or whether it should always choose the shortest path to the programmed position (even without M126). Examples:

Actual position	Nominal position	Traverse
350°	10°	–340°
10°	340°	+330°

Behavior with M126

With **M126**, the control will move a rotary axis, whose display is reduced to values less than 360°, on the shortest path of traverse. Examples:

Actual position	Nominal position	Traverse
350°	10°	+20°
10°	340°	–30°

Effect

M126 becomes effective at the start of the block.

To cancel **M126**, enter **M127**. At the end of program, **M126** is automatically canceled.

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

```
N50 M94*
```

Example: Reduce the display of the C axis

```
N50 M94  C*
```

Example: Reduce the display of all active rotary axes and then move the tool in the C axis to the programmed value

```
M50 G00  C+180 M94*
```

Effect

M94 is effective only in the NC block where it is programmed.

M94 becomes effective at the start of the block.

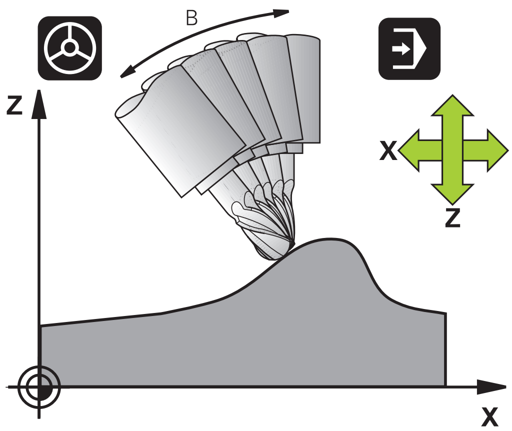
Retain position of tool tip when positioning tilted axes (TCPM): M128 (Option 9)

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 this offset. If the operator does not consider this deviation for the NC program, machining will occur with an offset.

Behavior with M128 (TCPM: Tool Center Point Management)

If the position of a controlled tilted axis changes in the NC program, the position of the tool tip to the workpiece remains the same.



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 program a feed rate at which the control will carry out the compensation movements in the linear axes.

If you want to change the position of the tilting axis with the handwheel during the program run, use **M128** along with **M118**. Superimposing handwheel positioning is implemented with active **M128**, depending on the setting in the 3D-ROT menu of **Manual operation** operating mode, in the active coordinate system or in the non-tilted coordinate system.

Programming notes:

- Before positioning axes with **M91** or **M92** and before a **T** block, reset the **M128** function
- To avoid contour damage, you must use only spherical cutters with **M128**.
- The tool length must refer to the spherical center of the Ball-nose cutter.
- If **M128** is active, 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, the control rotates the coordinate system accordingly. For example, if you rotate the C axis by 90 (through a positioning command or datum shift) and then program a movement in the X axis, the control executes the movement in the machine Y axis.

The control also transforms the preset, which has been shifted by the movement of the rotary table.

M128 with 3-D tool compensation

If you carry out a 3-D tool compensation with active **M128** and active radius compensation **G41**/**G42**, the control will automatically position the rotary axes for certain machine geometries (Peripheral milling).

Effect

M128 becomes effective at the start of block, **M129** at the end of block. **M128** is also effective in the manual operating modes and remains active even after a change of mode. The feed rate for the compensation movement will be effective until you program a new feed rate or until you cancel **M128** with **M129**.

Enter **M129** to cancel **M128**. The control also cancels **M128** if you select a new NC program in a program run operating mode.

Example: Feed rate of 1000 mm/min for compensation movements

```
N50 G01  G41 X+0  Y+38.5 IB-15 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 combination 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 Carry out the machining operation
5 At the end of the program, cancel **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 nominal position by a value greater than that defined by the machine tool builder, the control outputs an error message and interrupts program run.

Selecting tilting axes: M138

Standard behavior

The control performs **M128**, and **Tilt working plane** only for those axes that the machine tool builder has specified 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.

```
N50 G00  Z+100 G40  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 description of kinematics.

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

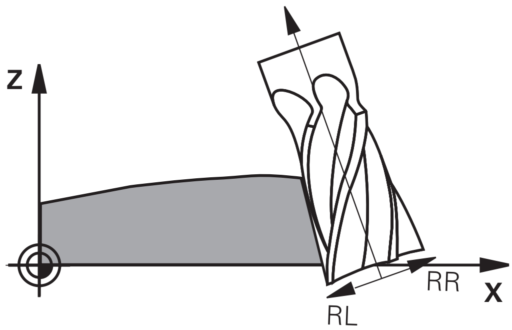
M144 becomes effective at the start of the block. **M144** does not work in connection with **M128** or the Tilt Working Plane function.

You can cancel **M144** by programming **M145**.

11.5 Peripheral Milling: 3-D radius compensation with M128 and radius compensation (G41/G42)

Application

With peripheral milling, the control displaces the tool perpendicularly to the direction of movement and perpendicularly to the tool direction by the total of the **DR** delta values (from the tool table and the **T** block). Use the **G41/G42** radius compensation to define the compensation direction (direction of movement Y+).



For the control to be able to reach the set tool orientation, you need to activate the **M128** function and subsequently the tool radius compensation. The control then positions the rotary axes automatically in such a way that the tool can reach the orientation defined by the coordinates of the rotary axes with the active compensation.

Further information: "Retain position of tool tip when positioning tilted axes (TCPM): M128 (Option 9)", Page 387

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 393

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 **Program run, single block** operating mode

You can define the tool orientation in a G01 block as described below.

Example: Definition of the tool orientation with M128 and the coordinates of the rotary axes

N10 G00 G90 X-20 Y+0 Z+0 B+0 C+0*	Pre-position
N20 M128*	Activate M128
N30 G01 G42 X+0 Y+0 Z+0 B+0 C+0 F1000*	Activate radius compensation
N40 X+50 Y+0 Z+0 B-30 C+0*	Position the rotary axis (tool orientation)

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

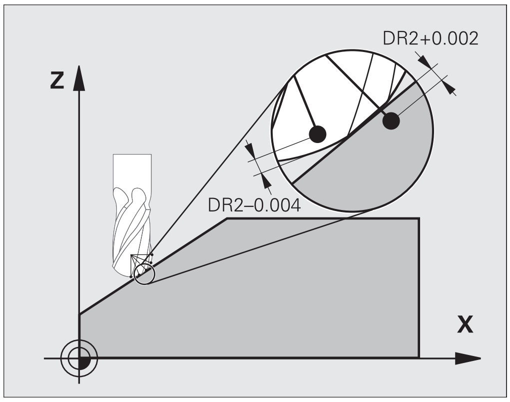
You have the following possibilities:

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



The effective sphere radius of a radius cutter deviates from the ideal form owing to the production process. The maximum form inaccuracy is defined by the machine tool builder. 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:

PGM MGT

- ▶ In the file manager open the path **TNC:\system \3D-ToolComp**

NEW FILE

- ▶ 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:

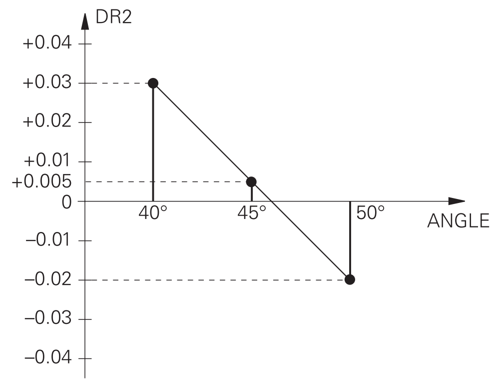
- **NR**: Consecutive line number
- **ANGLE**: Measured angle in degrees
- **DR2**: Radius deviation from the nominal value

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. If required, you can define additional offsets, such as a surface oversize, via DR2 in the **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:

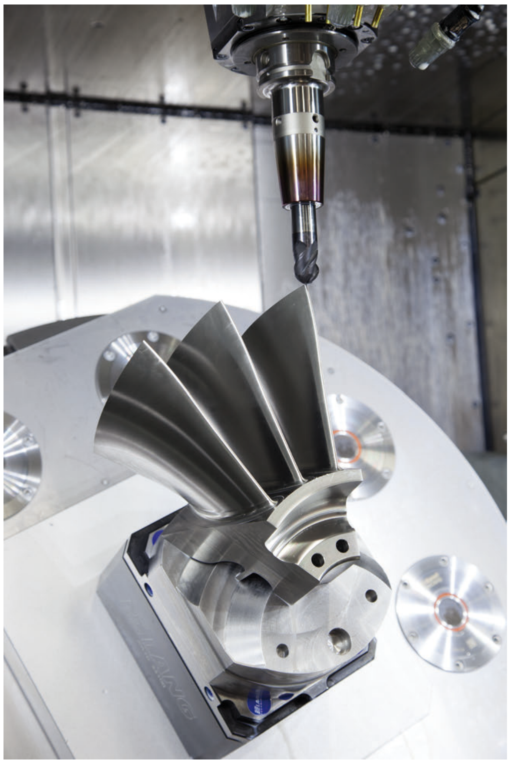
- NC program output at the south pole of the sphere requires tools measured on the tool tip
- NC program output at the center of the sphere requires tools measured on the tool center

11.6 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 real-time operating system HeROS 5 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 post processor generates a machine- and control-specific NC program, which can be processed by the CNC control. The post processor is adapted according to the machine tool and the control. The post processor 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 to five decimal places (option 23) 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 a precision of seven decimal places, as LN blocks are always calculated with a high accuracy, regardless of the setting of option 23
- 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 G32 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 G32.
- 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:

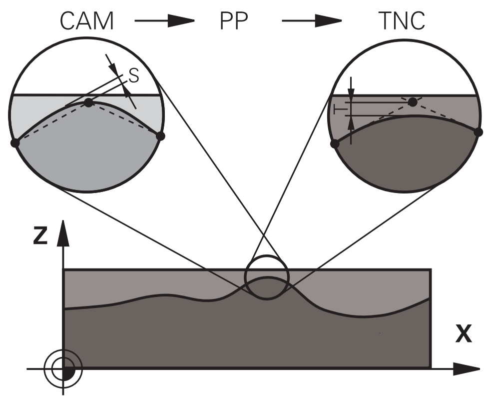
- In order to improve the structure of large NC programs, use the control's structuring function
  Further information: "Structuring NC programs", Page 188
- Use the control's commenting function in order to document NC programs
  Further information: "Adding comments", Page 184
- When machining holes 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 127
- 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 ; POSITION FEED RATE
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 G62, 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 G62. 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 G62: 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 an matching low tolerance in Cycle G62 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 G62: 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 G62 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 G62: 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 G62. 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 G62 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 radius cutters where the NC output is for the south pole of the sphere, choose a lower rotational 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 G62 **TOLERANCE** is available for influencing the behavior of CAM programs directly on the control. Please note the information describing the functioning of Cycle G62. 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

```
N340 G62 T0.05 P01 1 P02 3*
```

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

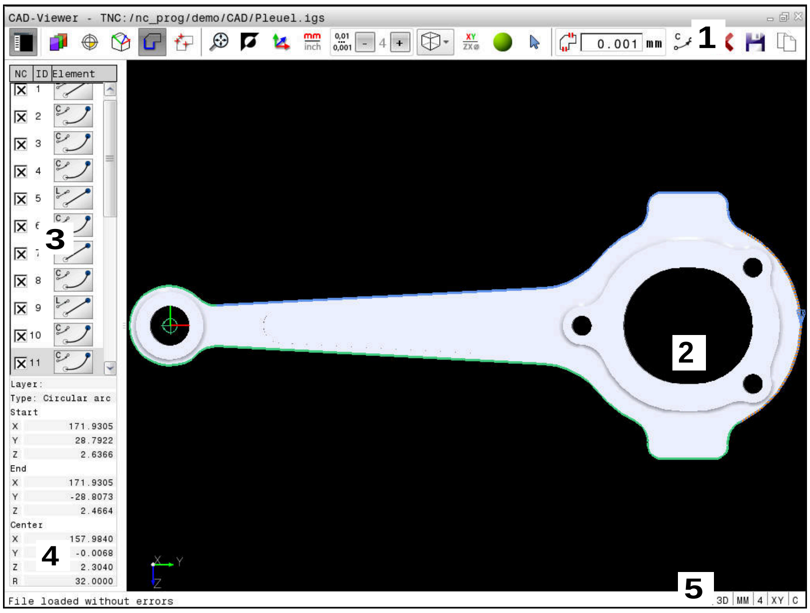
Data Transfer from CAD Files

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 Window element information
5 Status bar

File formats

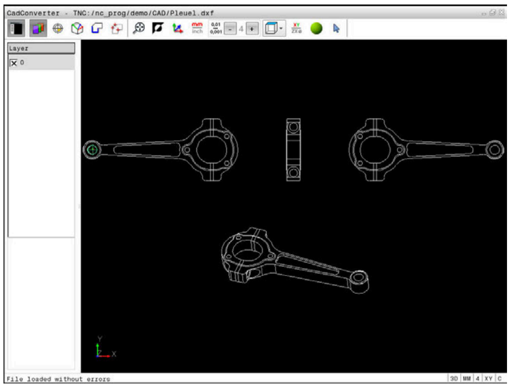
The **CAD-Viewer** enables you to open standardized CAD data formats directly on the control.
The control displays the following file formats:

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-Viewer (option 42)

Application

> If the control is set to ISO, the extracted contours or machining positions are nevertheless output as Klartext programs in **.H** conversational format.



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, 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 in 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 101
> - The control does not support binary DXF format. Save the DXF file in ASCII format in the CAD or drawing program.

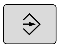
Using the CAD viewer

> 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.

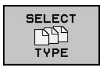
> If you are using a TNC 640 with touch control, you can replace some keystrokes with hand-to-screen contact.
>
> **Further information:** "Operating the Touchscreen", Page 473

Opening the CAD file

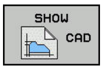


- Press the **Programming** key

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



- In order to see the soft key menu for selecting the file types to be displayed, press the **SELECT TYPE** soft key



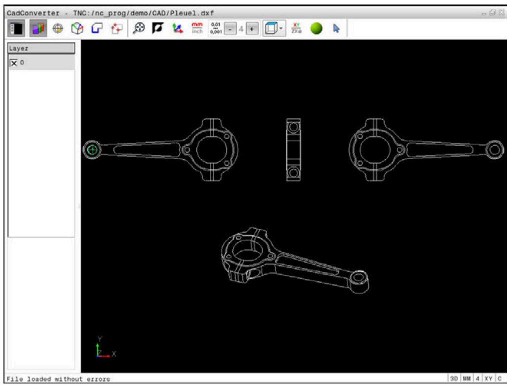
- To show all CAD files, press the **SHOW CAD** or **SHOW ALL** soft key
- Select the directory in which the CAD file is saved

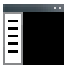
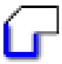
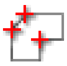
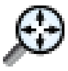
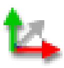
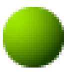
- Select the desired CAD file

- 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.

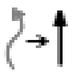
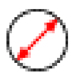
Basic settings

The basic settings specified below are selected using the icons in the toolbar.



Icon	Setting
	Show or hide the Window List view 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 view of the complete graphics
	Switch background color (black or white)
	Switch between 2-D and 3-D mode. The active mode is color-highlighted
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** and 5 decimal places with **inch** as unit of measure
	Switch between various view of the model e.g. **Top**
XY ZX	Select a contour for a turning operation. The active machining is color-highlighted (Option #50)
	Activate 3-D drawing wire model
	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

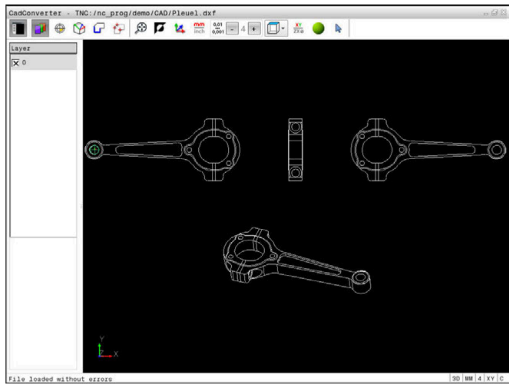
The following icons are displayed by the control only in certain modes.

Icon	Setting
	The most recent step is undone.
	Contour assumption 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
	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 assumption mode: Specify whether the control should display the tool path as a dashed line during selection of machining positions
	Path optimization mode: The control optimizes the tool traverse movement to give the shortest traverse movements between the machining positions. Optimization is reset with repeated actuations
	Hole position mode: The control opens a pop-up window in which you can filter bore holes (full circles) by size

Operating notes:

- Set the correct unit of measure, since the CAD file does not contain any such information.
- When generating NC programs for previous control models, you must limit the resolution to three decimal places. In addition, you must remove the comments that the **CAD-Viewer** inserts into the contour program.
- The control displays the active basic settings in the status bar of the screen.

Setting layers



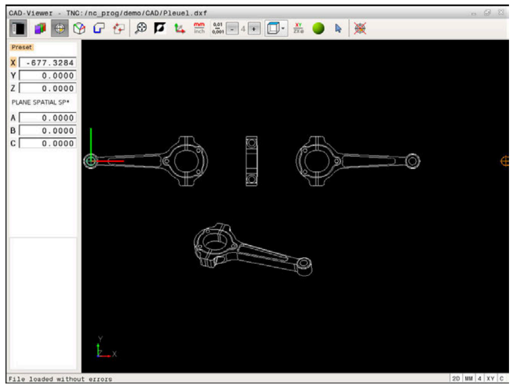
CAD files usually contain several layers. The designer uses these layers to create groups of various types of elements, e.g. the actual workpiece contour, dimensions, auxiliary and design lines, shadings, and texts.

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 adjust the orientation of the coordinate system, if required.
  Further information: "Adjusting the orientation of the coordinate system", Page 411

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 adjust the orientation of the coordinate system, if required.
  Further information: "Adjusting the orientation of the coordinate system", Page 411

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.

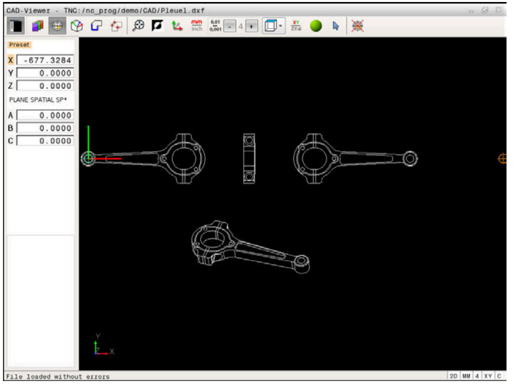
Adjusting the orientation of the coordinate system

The position of the coordinate system is defined by the orientation of the axes.

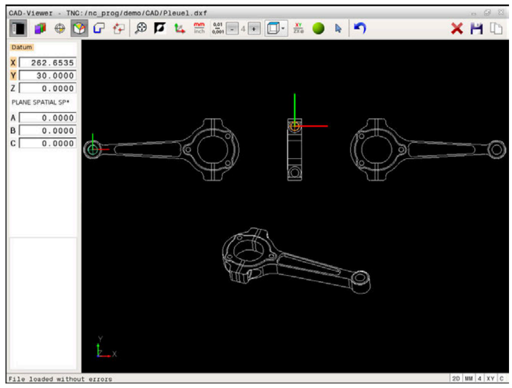
- ▶ The preset has already been set
- ▶ Left-click an element that is in the positive X direction
- > The control aligns the X axis and changes the angle in C.
- > The control colors the list view orange if the defined angle does not equal 0.
- ▶ Left-click an element that is approximately in the positive 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 410

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 adjust the orientation of the coordinate system, if required.
  Further information: "Adjusting the orientation of the coordinate system", Page 415

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 adjust the orientation of the coordinate system, if required.
  Further information: "Adjusting the orientation of the coordinate system", Page 415

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

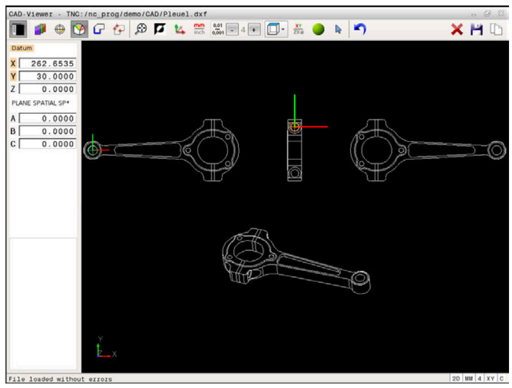
The position of the coordinate system is defined by the orientation of the axes.

- ▶ 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 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.

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

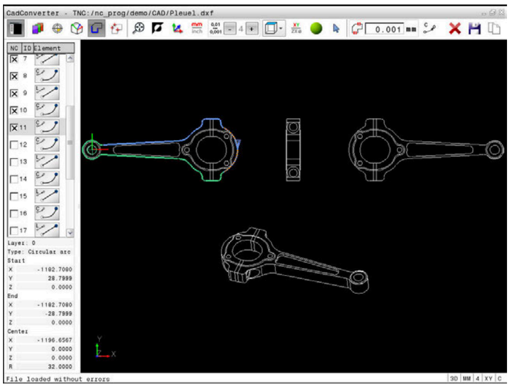
In the Element Information window, the control shows how far the datum you have chosen is located from the workpiece preset.



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

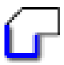
- Confirm the entry
- > The control saves the contour program to the selected directory.
- If you want to select more contours, press the Cancel Selected Elements soft key and select the next contour as described above

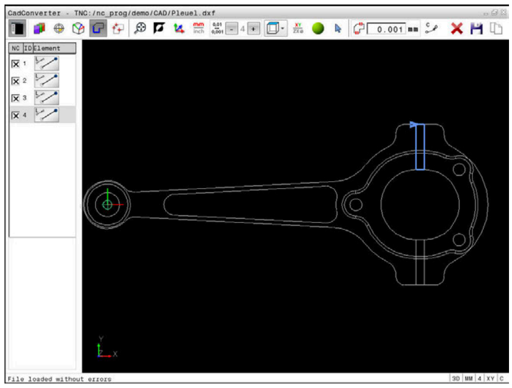
Operating notes:

- 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:



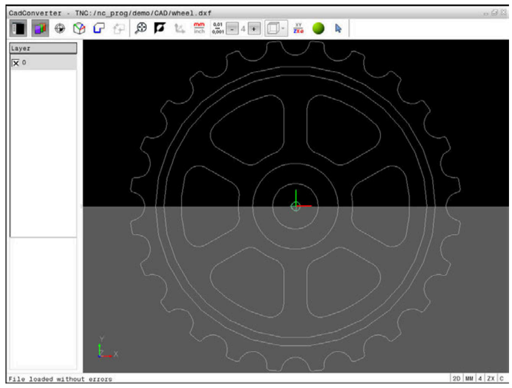


- ▶ The Graphics window is active for the contour selection
- ▶ To select the starting point, select an element or the intersection between two elements (using the **+** icon)
- ▶ Select the next contour element by clicking it with the mouse
- > The control displays the machining sequence as a dashed 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.

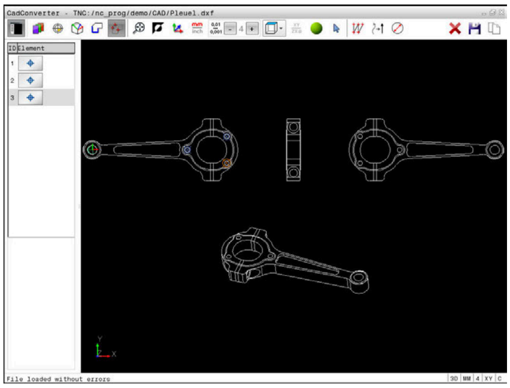
- ▶ Select the mode for choosing a turning contour
- > The control shows only the selectable elements above the rotation center.
- ▶ Select the desired contour elements with the left mouse button
- > The control displays the selected contour elements in blue and shows the selected elements with a symbol (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 407

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 421
- 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 with the mouse area", Page 422
- 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 423

Selecting the file type

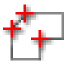
The following file types are available:

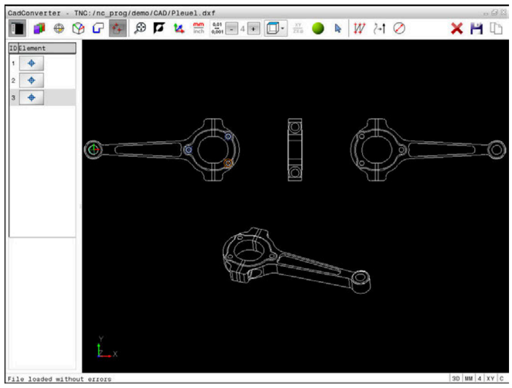
- Point table (**.PNT**)
- Klartext conversational language program (**.H**)

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.

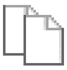
Single selection



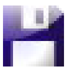


- ▶ Select the mode for choosing a machining position
- > The Graphics window is active for position selection.
- ▶ To select a machining position, click the element with the mouse
- > The control displays the element in orange.
- > If the shift key is pressed at the same time, the control indicates 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



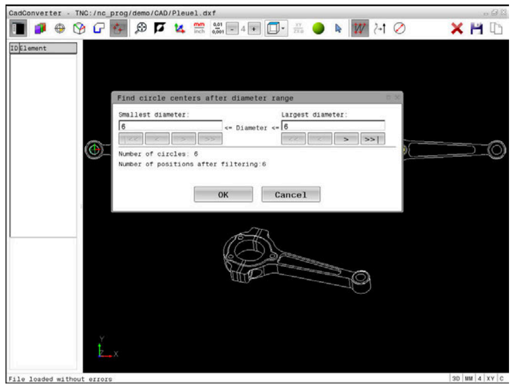
- ▶ Alternative: Save the selected machining positions in a point file
- > The control displays a pop-up window in which you can select the target directory, a file name, and the file type.

ENT

- ▶ Confirm the entry
- > The control saves the contour program to the selected directory.

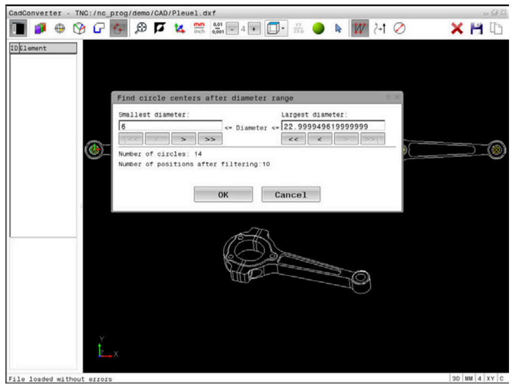
- ▶ If you want to select more machining positions, press the Cancel Selected Elements icon and select as described above

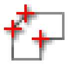
Rapid selection of hole positions with the mouse area



- 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 424
- > 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.
- 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

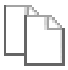




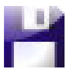
- Select the mode for choosing machining positions
- > The Graphics window is active for position selection.

- 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 424
- > 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.

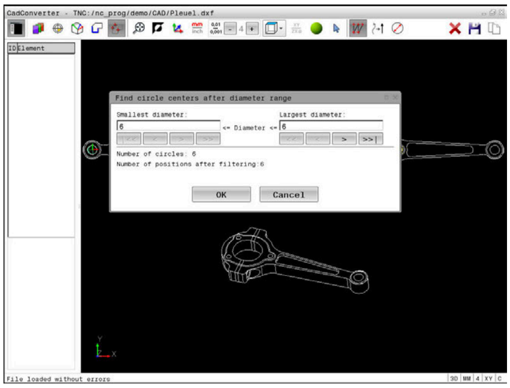
ENT

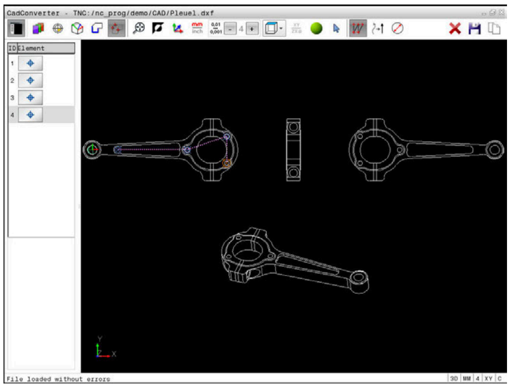
- Confirm the entry
- > The control saves the contour program to the selected directory.

- 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 have the tool paths displayed by clicking the **SHOW TOOL PATH** icon.

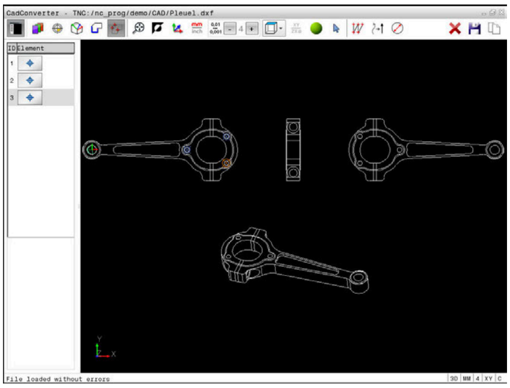
Further information: "Basic settings", Page 407

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

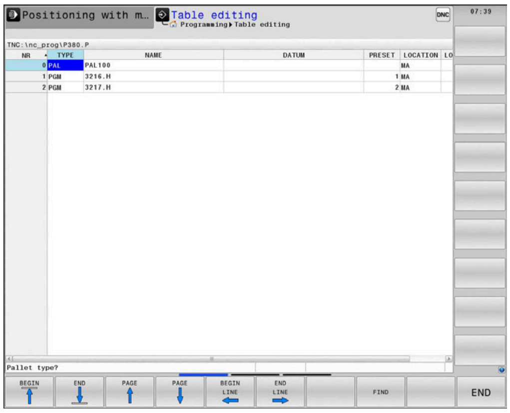


13

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 431

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 G62 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: 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:

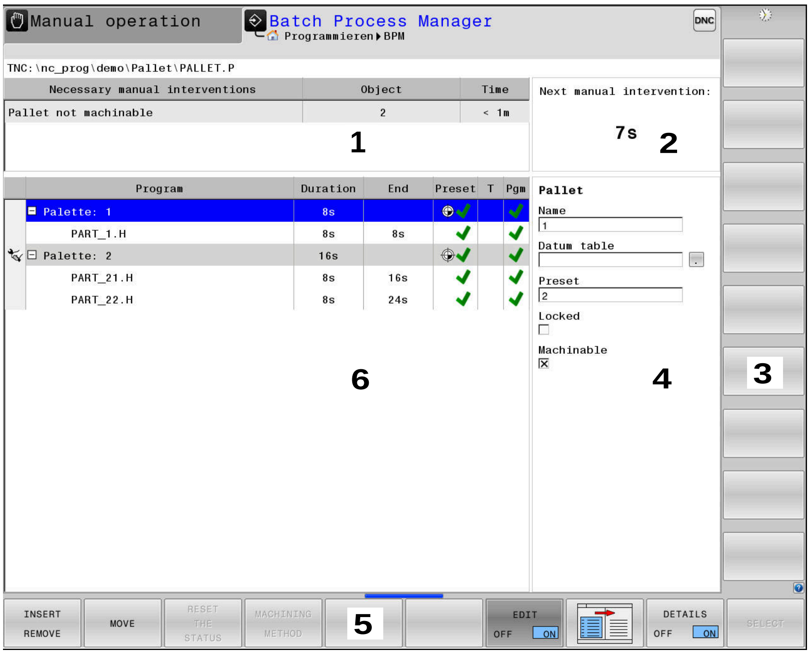
- **Programming**
- **Program run, single block**
- **Program run, full sequence**

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**, **Fixture** or **Program**
Program	Name or path of the **Pallet**, **Fixture** or **Program**
Duration	Run time in seconds This column is only shown if your machine has a 19-inch screen.
End Time	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**, **Fixture** and **Program** is shown by means of icons in the first column.

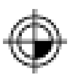
The icons have the following meanings:

Icon	Meaning
	**Pallet**, **Fixture** or **Program** is locked
	**Pallet** or **Fixture** 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 failed, e.g. because of expired tool life
	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.

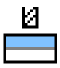
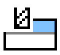
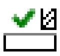
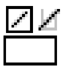
Operating notes:

- 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 shown in the **Batch Process Manager** only 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:

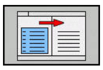
- Press the **Screen layout** key

BPM
- Press the 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
CANCEL THE MARKING	Cancel marking
INSERT BEFORE	Insert a new **Pallet**, **Fixture** or **Program** before the cursor position
INSERT AFTER	Insert a new **Pallet**, **Fixture** 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
TOOL MANAGEMENT	Open the Expanded tool management
INTERNAL STOP	Interrupt machining

Operating notes:

- The **TOOL MANAGEMENT** 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.

> **i** 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**
  - **Fixture**
  - **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.

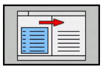
> Operating notes:
>
> - If a job list is selected in the **Program run, single block** or **Program run, full sequence** operating mode, it is not possible to edit the job list in the **Programming** operating mode.
> - The possibilities of changing a job list during machining are limited, because the control defines a protected area.
> - NC programs in the protected area are shown in light gray.

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**:

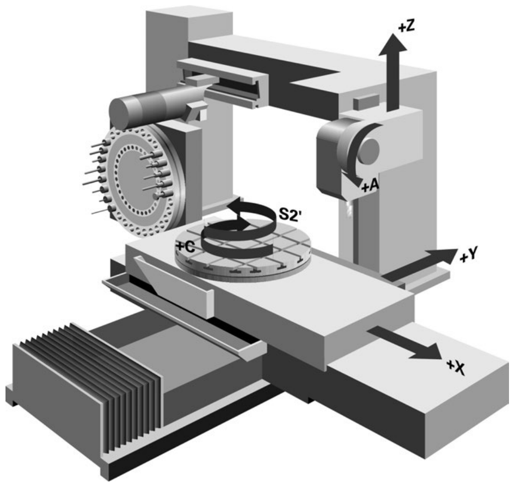
- ▶ Open the desired job list
- ▶ Press the **EDIT** soft key
- ▶ Place the cursor on the desired line, e.g. **Program**
- > The control displays the selected line in blue.
- ▶ Press the **MOVE** soft key
- ▶ Press the **TAG** soft key
- > The control highlights the line in which the cursor is positioned.
- ▶ 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



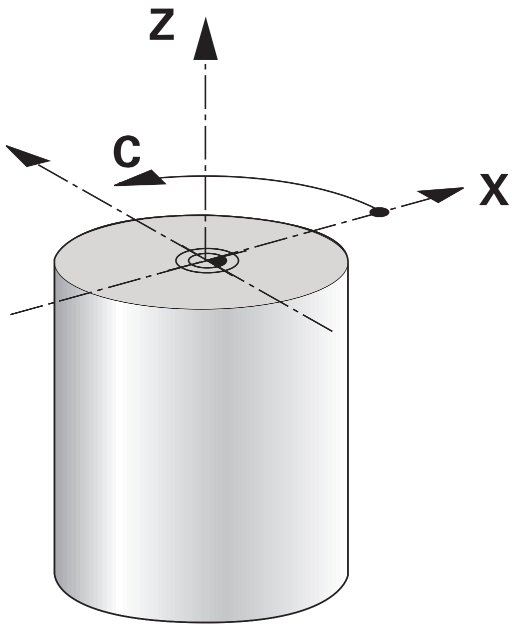
Special types of milling machines allow performing both milling and drilling operations. A workpiece can thus be machined completely on one machine without rechucking, even if complex milling and turning applications are required.

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, different geometric descriptions to those for milling or drilling tools are considered. 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.

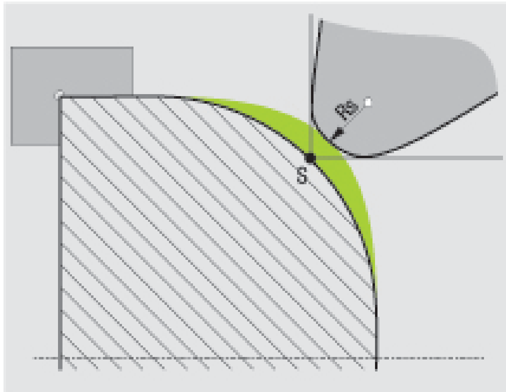
Further information: "Inclined turning", Page 464

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

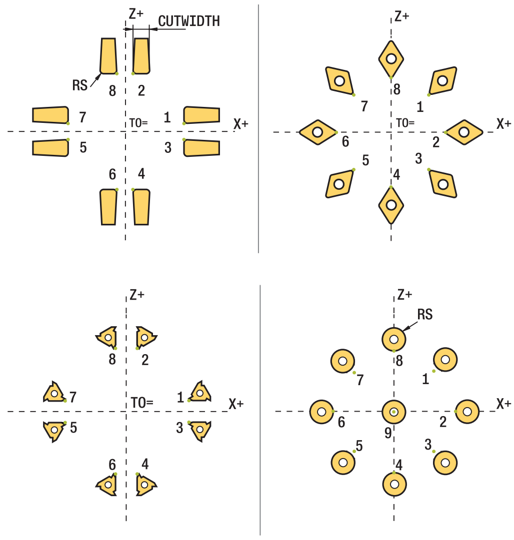


The tip of a lathe tool has a certain radius (**RS**). When machining tapers, chamfers and radii, this results in distortions on the contour because the programmed traverse paths refer to the theoretical tool tip S. TRC prevents the resulting deviations.

In the turning cycles the control automatically carries out tool radius compensation. In specific traversing blocks and within programmed contours, activate TRC with **G41** or **G42**.

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.

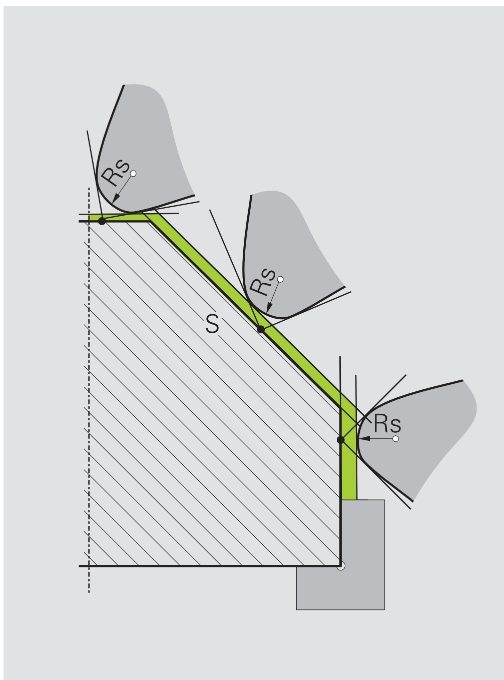
  The control can also run tool tip radius compensation during inclined processing.

  Active miscellaneous functions limit the possibilities here:

  - With **M128** tool-tip radius compensation is possible only in combination with machining cycles
  - **M144** or also allows tool tip radius compensation with all traversing blocks, e.g. with **G41/G42**

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.



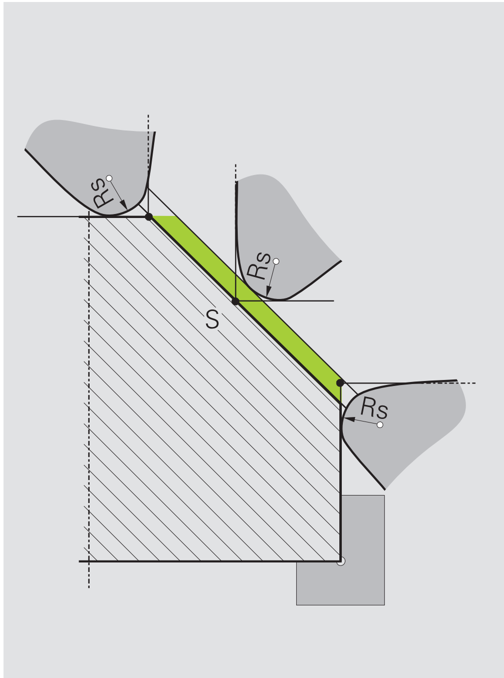
Virtual tool tip

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



14.2 Basic functions (option 50)

Switching between milling/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**.

The control shows an icon in the status display when the turning mode is active

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 prepositioned 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 318

Entering the operating mode:

SPEC FCT
- Show the soft-key row with special functions

TURNING PROGRAM FUNCTIONS
- Press the **TURNING PROGRAM FUNCTIONS** soft key

BASIC FUNCTIONS
- Press the **BASIC FUNCTIONS** soft key

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:

- Enter **"** quotation marks

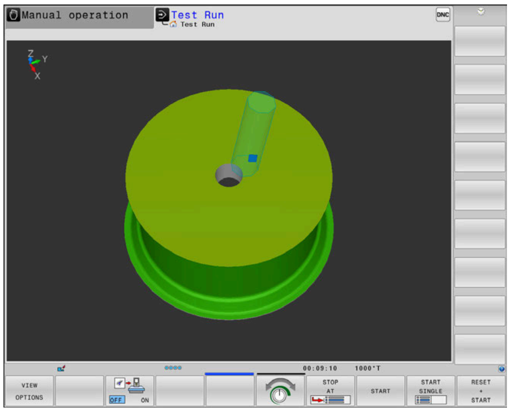
SELECT
- Press the **SELECT KINEMATICS** soft key

Example

11 FUNCTION MODE TURN "AC_TABLE"	Activate turning mode
N120 FUNCTION MODE TURN*	Activate turning mode
N130 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.

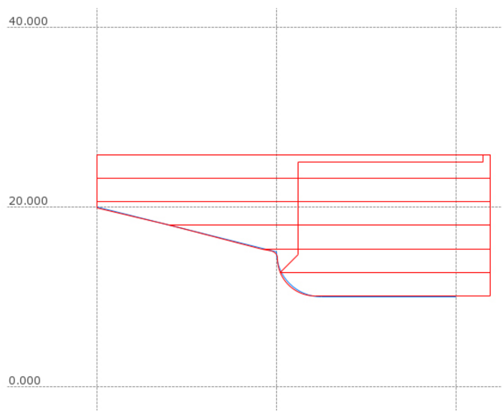


> The machining times determined using the graphic simulation do not correspond to the actual machining times. Reasons for this during combined milling-turning operations include the switching of operating modes.

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 199



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.

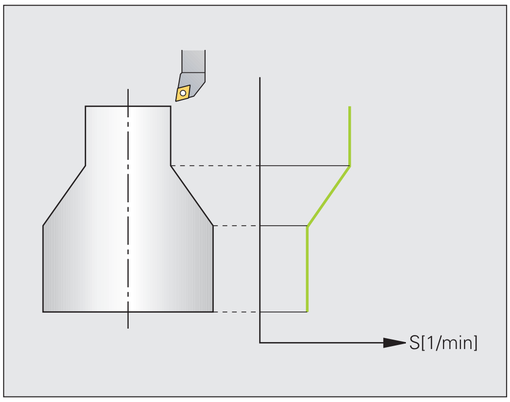
Even if the turning operation takes place in a two-dimensional plane (Z and X coordinates), you have to program the Y values for a rectangular blank in the definition of the workpiece blank.

Example. Rectangular blank

%LT 200 G71 *	
N10 G30 G18 X+0 Y-1 Z-50*	Define the workpiece blank for graphic workpiece simulation
N20 G31 G90 X+87 Y+1 Z+2*	
N30 T301*	Tool call
N40 G00 G40 G90 Z+250*	Retract the tool in the spindle axis at rapid traverse
N50 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 (obligatory)
- 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:

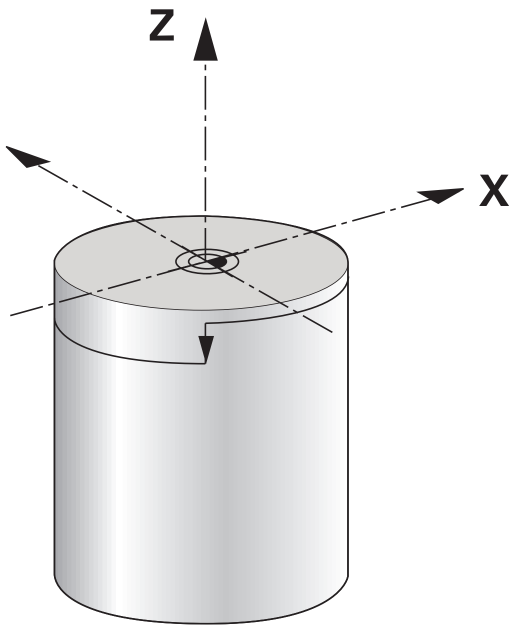
SPEC FCT	▶ Show the soft-key row with special functions
TURNING PROGRAM FUNCTIONS	▶ Press the **TURNING PROGRAM FUNCTIONS** soft key
FUNCTION TURNDATA	▶ Press the **FUNCTION TURNDATA** soft key
TURNDATA SPIN	▶ Press the **TURNDATA SPIN** soft key.
VCONST: ON	▶ Select the function for speed entry: Press the **VCONST:** soft key

> Cycle G800 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

N30 FUNCTION TURNDATA SPIN VCONST:ON VC:100 GEARRANGE:2*	Definition of a constant cutting speed in gear range 2
N30 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

%LT 200 G71 *	
N40 G00 G40 G90 X+102 Z+2*	Movement at rapid traverse
...	
N30 G01 X+87 F200*	Movement at a feed rate of 200 mm/min
N40 M136*	Feed rate in millimeters per revolution
N50 G01 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 **FUNCTION TURNDATA CORR** 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 **T** deactivates compensation again. When you exit the NC program (e.g. with PGM MGT), the control automatically resets the compensation values.

When you enter the **TURNDATA CORR FUNCTION** you can specify the effect of the tool compensation with a soft key:

- **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

Define the tool compensation:

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 CORR
- Press the **TURNDATA CORR** soft key.

Example

```
N210 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

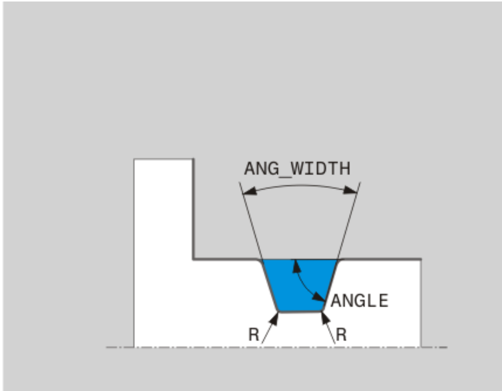
RECESS/ UNDERCUT
- Press the **RECESS/ UNDERCUT** soft key

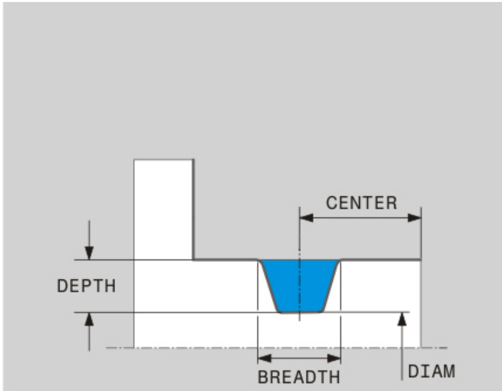
GRV
- Press the **GRV** (recess) or **UDC** (undercut) soft key

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

```
N30 G01  X+40 Z+0*
N40 G01  Z-30*
N50 GRV RADIAL  CENTER-15 DEPTH-5  BREADTH10 CHF1
    FAR_CHF1*
N60 G01  X+60*
```

Programming undercutting

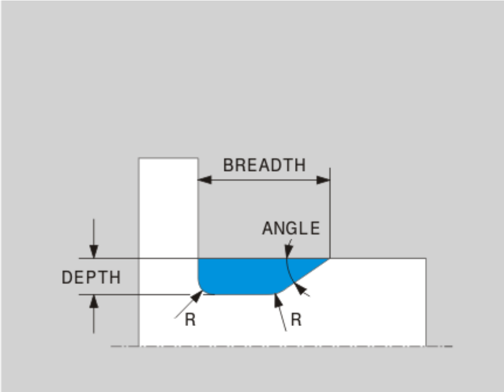
Undercutting is usually required for the flush connection of counterparts. In addition undercutting can help to reduce the notch effect at corners. Threads and fits are often machined with an undercut. You have various contour elements for defining the different undercuts:

- **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



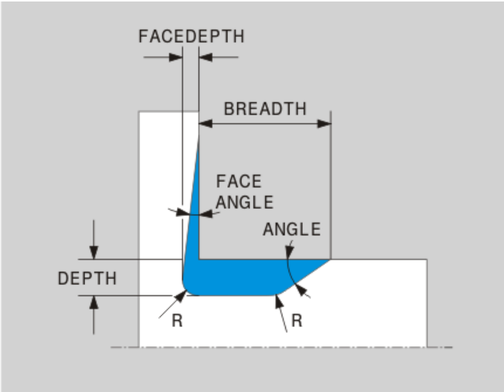
Input parameters	Application	Input
R	Corner radius of both inner corners	Optional
DEPTH	Undercut depth	Optional
BREADTH	Width of undercut	Optional
ANGLE	Undercut angle	Optional

Example: Undercut with depth = 2, width = 15

```
N30 G01  X+40 Z+0*
N40 G01  Z-30*
N50 UDC TYPE_E R1 DEPTH2 BREADTH15*
N60 G01  X+60*
```

Undercut DIN 509 UDC TYPE_F

Input parameters in undercut DIN 509 UDC TYPE_F



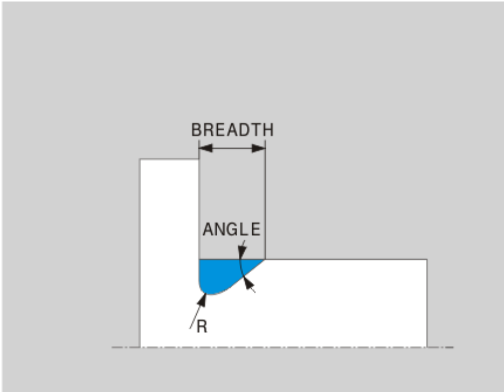
Input parameters	Application	Input
R	Corner radius of both inner corners	Optional
DEPTH	Undercut depth	Optional
BREADTH	Width of undercut	Optional
ANGLE	Undercut angle	Optional
FACEDEPTH	Depth of face	Optional
FACEANGLE	Contour angle of face	Optional

Example: Undercut form F with depth = 2, width = 15, depth of face = 1

```
N30 G01  X+40 Z+0*
N40 G01  Z-30*
N50 UDC TYPE_F R1 DEPTH2 BREADTH15 FACEDEPTH1*
N60 G01  X+60*
```

Undercut DIN 509 UDC TYPE_H

Input parameters in undercut DIN 509 UDC TYPE_H



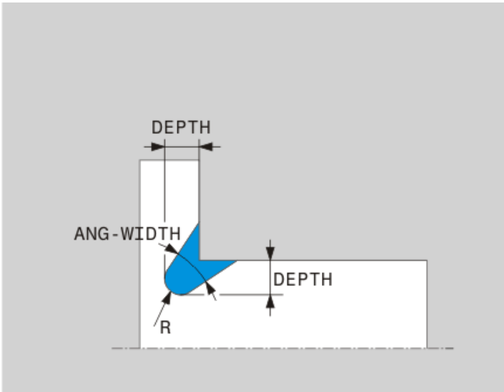
Input parameters	Application	Input
R	Corner radius of both inner corners	Required
BREADTH	Width of undercut	Required
ANGLE	Undercut angle	Required

Example: Undercut form H with depth = 2, width = 15, angle = 10°

```
N30 G01  X+40 Z+0*
N40 G01  Z-30*
N50 UDC TYPE_H R1 BREADTH10 ANGLE10*
N60 G01  X+60*
```

Undercut UDC TYPE_K

Input parameters in undercut UDC TYPE_K



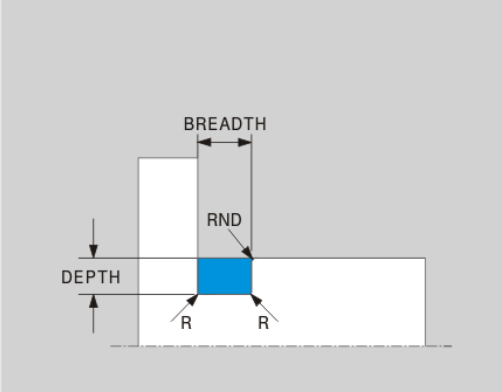
Input parameters	Application	Input
R	Corner radius of both inner corners	Required
DEPTH	Undercut depth (paraxi-al)	Required
ROT	Angle to longitudinal axis (default: 45°)	Optional
ANG_WIDTH	Opening angle of under-cut	Required

Example: Undercut form K with depth = 2, width = 15, opening angle = 30°

```
N30 G01  X+40 Z+0*
N40 G01  Z-30*
N50 UDC TYPE_K R1 DEPTH3 ANG_WIDTH30*
N60 G01  X+60*
```

Undercut UDC TYPE_U

Input parameters in undercut UDC TYPE_U



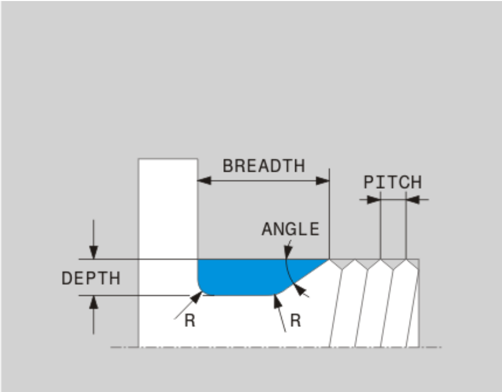
Input parameters	Application	Input
R	Corner radius of both inner corners	Required
DEPTH	Undercut depth	Required
BREADTH	Width of undercut	Required
RND / CHF	Curve / chamfer of outer corner	Required

Example: Undercut form U with depth = 3, width = 8

```
N30 G01  X+40 Z+0*
N40 G01  Z-30*
N50 UDC TYPE_U R1 DEPTH3 BREADTH8 RND1*
N60 G01  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

```
N30 G01  X+40 Z+0*
N40 G01  Z-30*
N50 UDC THREAD PITCH2*
N60 G01  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

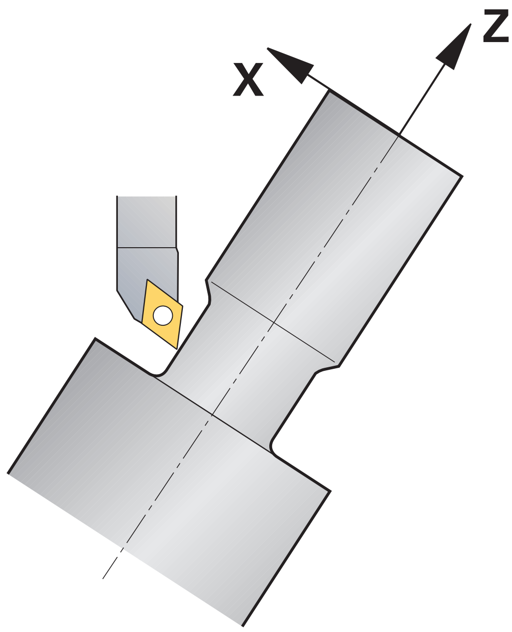
Inclined turning

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**

If the turning cycles are executed with **M144**, or **M128**, the angles of the tool to the contour change. The control automatically takes these modifications into account and thus also monitors the machining in an inclined state.



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.

...	
N10 M144*	Activate inclined machining
N20 G00 A-25 G40*	Position swivel axis
N30 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	
N40 G00 X+165 Y+0 G40*	Pre-positioning the tool
N50 G00 Z+2 G40*	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 **G41**/**G42**, is not possible. If you activate inclined machining via **M144** then this limitation does not apply.

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!

Use **FUNCTION MODE TURN** to select a kinematic model prepared by the machine tool builder, which is necessary for the use of facing slide. With this kinematic model, the controls executes the programmed X-axis movements of the facing slide as U-axis movements if the **FACING HEAD** function is active. This automatism does not work if the **FACING HEAD** function is inactive and in **Manual operation** mode, which means that **X**-movements (programmed or axis key) are executed in the X axis. In this case, the facing slide has to be moved with the U axis. There is a danger 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 the **Manual operation** mode, move the facing slide with the **U** axis key.
- As 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

N50 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

N70 FACING HEAD POS*	Activating without clearance height
N70 FACING HEAD POS HEIGHT+100 F1000*	Activating with positioning to clearance height Z+100 at rapid traverse 1000

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

N70 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.

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 **EXIT LEARNING** 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

Operating the Touchscreen

15.1 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)

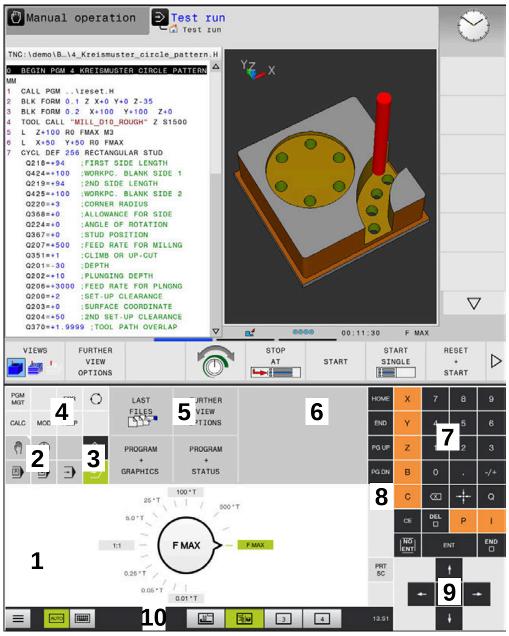
7 Numerical input and axis selection

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

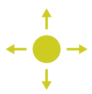
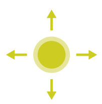
The following keys, for example, can easily be replaced by hand gestures:

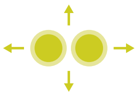
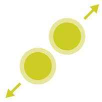
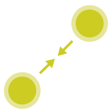
Key	Function	Gesture
	Switch between operating modes	Tap on the operating mode in the header
	Shift the soft-key row	Swipe horizontally over the soft-key row
	Soft-key selection keys	Tap on the function in the touchscreen

15.2 Gestures

Overview of possible gestures

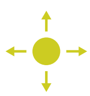
The screen of the control is multi-touch capable. That means that it can distinguish various gestures, including with two or more fingers at once.

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

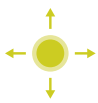
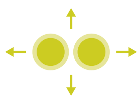
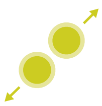
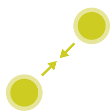
Operating the simulation

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.

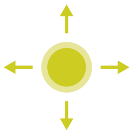
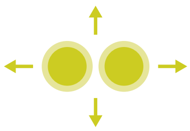
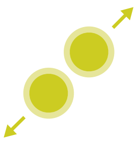
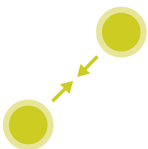
To be able to use all applications, first use the icon to select the desired function:

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

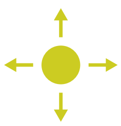
The control supports the following gestures:

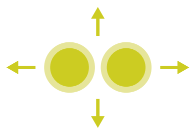
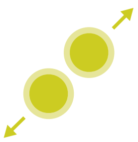
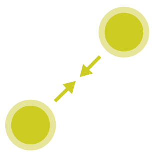
Symbol	Gesture	Function
	Tap on an element	Show element information Specify the workpiece preset
	Double-tap on the background	Set the graphic or 3-D model to its original size

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

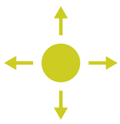
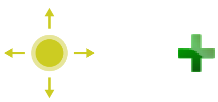
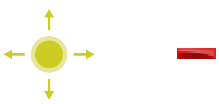
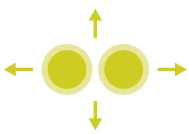
The control supports the following gestures:

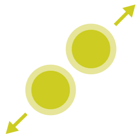
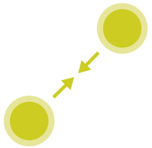
Symbol	Gesture	Function
	Tap on an element	Select element
	Tap on an element in the list-view window	Select or deselect an element
	Activate **Add** and tap on an element	Part, shorten, or lengthen and element
	Activate **Remove** and tap on an element	Deselect an element
	Double-tap on the background	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

16

Tables and Overviews

16.1 System data

List of D18 functions

With the **D18** 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 **D18** 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 **D18** 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.
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
		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

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		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)
		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)

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		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)
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
		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

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		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 range. If no gear ranges 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
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 participates 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/s^3
		5	-	Max. jerk at high speeds (**MP_maxPathJerkHi**) in m/s^3
		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 "..."**.
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: D18 functions

The following table lists the D18 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)
8	Pocket no.	P2	2)

No.	IDX	Contents	Replacement function
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 FN26: 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 26/28 Read out the 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 / FN 28 or SQL to read out the table cell

16.2 Overview tables

Miscellaneous functions

M	Effect	Effective at block Start	End	Page
M0	Program STOP/Spindle STOP/Coolant OFF		■	216
M1	Optional program run STOP/Spindle STOP/Coolant OFF		■	216
M2	Stop program/Spindle STOP/Coolant OFF/ CLEAR status display (depending on machine parameter)/Return jump to block 1		■	216
M3 M4 M5	Spindle ON clockwise Spindle ON counterclockwise Spindle STOP	■ ■	 ■	216
M6	Tool change/STOP program run (depending on machine parameter)/Spindle STOP		■	216
M8 M9	Coolant ON Coolant OFF	■	 ■	216
M13 M14	Spindle ON clockwise/Coolant ON Spindle ON counterclockwise/Coolant on	■ ■		216
M30	Same function as M2		■	216
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	■		217
M92	Within the positioning block: Coordinates are referenced to a position defined by machine manufacturer, e.g. tool change position	■		217
M94	Reduce the rotary axis display to a value below 360°	■		386
M97	Machine small contour steps		■	220
M98	Machine open contours completely		■	221
M99	Blockwise cycle call		■	Cycles Manual
M101 M102	Automatic tool change with replacement tool if maximum tool life has expired Reset M101		■ ■	124
M107 M108	Suppress error message for replacement tools with oversize Reset M107		■ ■	124
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	■ ■	 ■	223
M116 M117	Feed rate in mm/min on rotary axes Reset M116	■	 ■	384
M118	Superimpose handwheel positioning during program run	■		226
M120	Pre-calculate the radius-compensated contour (LOOK AHEAD)	■		224
M126 M127	Shorter-path traverse of rotary axes Reset M126	■	 ■	385
M128 M129	Maintaining the position of the tool tip when positioning with tilted axes (TCPM) Reset M128	■	 ■	387

M	Effect	Effective at block	Start	End	Page
M130	Within the positioning block: Points are referenced to the untilted coordinate system		■		219
M136 M137	Feed rate F in millimeters per spindle revolution Reset M136		■		223
M138	Selection of tilted axes		■		390
M140	Retraction from the contour in the tool-axis direction		■		228
M143	Delete basic rotation		■		231
M144 M145	Compensating the machine’s kinematic configuration for ACTUAL/NOMINAL positions at end of block Reset M144		■	 ■	391
M141	Suppress touch probe monitoring		■		230
M148 M149	Automatically retract tool from the contour at an NC stop Reset M148		■	 ■	232

User functions

User functions		
Short description	■	Basic version: 3 axes plus closed-loop spindle
	■	Fourth NC axis plus auxiliary axis
		or
	□	8 additional axes or 7 additional axes plus 2nd spindle
	■	Digital current and speed control
Program entry	In HEIDENHAIN conversational format and DIN/ISO	
Position entry	■	Nominal positions for lines and arcs in Cartesian coordinates or polar coordinates
	■	Incremental or absolute dimensions
	■	Display and entry in mm or inches
Tool compensation	■	Tool radius in the working plane and tool length
	■	Radius-compensated contour look-ahead for up to 99 NC blocks (M120)
	2	Three-dimensional tool-radius compensation for changing tool data without having to recalculate an existing NC program
Tool tables	Multiple tool tables with any number of tools	
Constant contour speed	■	With respect to the path of the tool center
	■	With respect to the cutting edge
Parallel operation	Creating an NC program with graphical support while another NC program is being run	
3-D machining (Advanced Function Set 2)	**2**	Motion control with minimum jerk
	2	3-D tool compensation through surface-normal vectors
	2	Using the electronic handwheel to change the angle of the swivel head during program run without affecting the position of the tool center point (tool tip or center of sphere) (TCPM = **T**ool **C**enter **P**oint **M**anagement)
	2	Keeping the tool normal to the contour
	2	Tool radius compensation perpendicular to traversing and tool direction
Rotary table machining (Advanced Function Set 1)	**1**	Programming of cylindrical contours as if in two axes
	1	Feed rate in distance per minute

User functions		
Contour elements	■	Straight line
	■	Chamfer
	■	Circular path
	■	Circle center
	■	Circle radius
	■	Tangentially connected arc
	■	Rounded corners
Approaching and departing the contour	■	Via straight line: tangential or perpendicular
	■	Via circular arc
FK free contour programming	■	FK free contour programming in HEIDENHAIN conversational format with graphic support for workpiece drawings not dimensioned for NC
Program jumps	■	Subprograms
	■	Program section repeat
	■	Any desired NC program as subprogram
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
	■	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

User functions	
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**Test Run** ■ 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

16.3 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:		
- **FU** (feed per revolution mm/1)	- –	- X
- **FZ** (tooth feed rate)	- –	- X
- **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

Function	TNC 640	iTNC 530
Q parameter programming:		
■ D15: PRINT	■ –	■ X
■ D25: PRESET	■ –	■ X
■ D29: PLC LIST	■ X	■ –
■ D31: RANGE SELECT	■ –	■ X
■ D32: PLC PRESET	■ –	■ X
■ D37: EXPORT	■ X	■ –
■ Write to LOG file with **D16**	■ X	■ –
■ Displaying parameter contents in the additional status display	■ X	■ –
■ **SQL** functions for writing and reading tables	■ X	■ –
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	■ –

Function	TNC 640	iTNC 530
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
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	■ –

Function	TNC 640	iTNC 530
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
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**	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
225 **ENGRAVING**	X	X
230 **MULTIPASS MILLING** (recommended: Cycle 233)	–	X
231 **RULED SURFACE**	–	X
232 **FACE MILLING**	X	X
233 **FACE MILLING**	X	–
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
275 **TROCHOIDAL SLOT**	X	X
276 **THREE-D CONT. TRAIN**	X	X
285 **DEFINE GEAR**	X, option 157	–

Cycle	TNC 640	iTNC 530
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	–
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 IMBALANCE**	X, option 50	–

Comparison: Touch probe cycles in the Manual operation and Electronic handwheel operating modesElectronic handwheel

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	■ 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 while the **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 while **CYCL CALL**, **SPEC FCT**, **PGM CALL** and **APPR/DEP** 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	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	■ Not available	■ 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 created when the system crashes	■ Available. A service file is automatically created when the system crashes
Find function:		
■ List of words recently searched for	■ Not available	■ Available
■ Show elements of active block	■ Not available	■ Available
■ Show list of all available NC blocks	■ Not available	■ Available
Starting the search function with the up/down arrow keys when highlighted	Works up to max. 100000 NC blocks, can be set via configuration datum	No limitation regarding program length
Programming graphics:		
■ True-to-scale display of grid	■ Available	■ Not available
■ Editing contour subprograms in SLII cycles with **AUTO DRAW ON**	■ With error messages, in the main program the cursor is positioned on the **CYCL CALL** NC block	■ With error messages, the cursor is positioned on the NC block in the contour subprogram that caused the error
■ Moving the zoom window	■ Repeat function not available	■ Repeat function available

Function	TNC 640	iTNC 530
Programming minor axes:		
■ Syntax **FUNCTION PARAXCOMP**: Define the behavior of the display and the paths of traverse	■ Available	■ Not available
■ Syntax **FUNCTION PARAXMODE**: Define the assignment of the parallel axes to be traversed	■ Available	■ Not available
Programming OEM cycles		
■ Access to table data	■ Via **SQL** commands and via **FN 17**/**FN 18** or **TABREAD-TABWRITE** functions	■ Via **FN 17**/**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 **%** 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

16.4 DIN/ISO Function Overview TNC 640

M functions	
M00 M01 M02	Program run STOP/Spindle STOP/Coolant OFF Optional program run STOP Program run STOP/Spindle/STOP/Coolant OFF/if nec. Clear status display (depending on machine parameter)/Return jump to block 1
M03 M04 M05	Spindle ON clockwise Spindle ON counterclockwise Spindle STOP
M06	Tool change/Program run STOP (depending on machine parameter)/Spindle STOP
M08 M09	Coolant ON Coolant OFF
M13 M14	Spindle ON clockwise/Coolant ON Spindle ON counterclockwise/Coolant on
M30	Same function as M02
M89	Free miscellaneous function or cycle call, modally effective (depending on machine parameter)
M99	Blockwise cycle call
M91 M92	Within the positioning block: Coordinates are referenced to machine datum Within the positioning block: Coordinates are referenced to position defined by machine tool builder, such as tool change position
M94	Reduce the rotary axis display to a value below 360°
M97 M98	Machine small contour steps Machine open contours completely
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
M116 M117	Feed rate for rotary axes in mm/min Reset M116
M118	Superimpose handwheel positioning during program run
M120	Pre-calculate radius-compensated contour (LOOK AHEAD)
M126 M127	Shorter-path traverse of rotary axes: Reset M126
M128 M129	Maintain position of the tool tip when positioning with tilted axes (TCPM) Reset M128
M130	Within the positioning block: Points are referenced to the untilted coordinate system
M140	Retraction from the contour in the tool-axis direction
M141	Suppress touch probe monitoring
M143	Delete basic rotation
M148 M149	Retract the tool automatically from the contour at NC stop Reset M148

G codes	
Tool movements	
G00	**Cartesian line in rapid traverse**
G01	**Cartesian line at feed rate**
G02	**Cartesian circle clockwise**
G03	**Cartesian circle CCW**
G05	**Cartesian circle**
G06	**Cartesian circle, tang. transit.**
G07*	**Cartesian line, paraxial**
G10	**Polar line in rapid traverse**
G11	**Polar line at feed rate**
G12	**Polar circle clockwise**
G13	**Polar circle counterclockwise**
G15	**Polar circle**
G16	**Polar circle, tang. transition**
Chamfer/Rounding/Approach contour/Depart contour	
G24*	**Chamfer with length R** with chamfer length R
G25*	**Corner rounding with radius R** with radius R
G26*	**Tangential approach** to a contour with radius R
G27*	**Tangential departure** from a contour with radius R
Tool definition	
G99*	**Tool definition** with tool number T, length L and radius R
Tool radius compensation	
G40	**Path of tool center** without tool radius compensation
G41	**Radius compensation left of path**
G42	**Radius compens. right of path**
G43	**Radius compensation: extend path** for G07
G44	**Radius compens.: shorten path** for G07
Blank form definition for graphics	
G30	**Workpiece blank def.: MIN point** (G17/G18/G19)
G31	**Workpiece blank def.: MAX point** (G90/G91)
Cycles for drilling, tapping and thread milling	
G200	**DRILLING**
G201	**REAMING**
G202	**BORING**
G203	**UNIVERSAL DRILLING**
G204	**BACK BORING**
G205	**UNIVERSAL PECKING**
G206	**TAPPING** with floating tap holder
G207	**RIGID TAPPING** without floating tap holder
G208	**BORE MILLING**
G209	**TAPPING W/ CHIP BRKG**
G240	**CENTERING**
G241	**SINGLE-LIP D.H.DRLNG**

G codes	
Cycles for drilling, tapping and thread milling	
G262	**THREAD MILLING**
G263	**THREAD MLLNG/CNTSNKG**
G264	**THREAD DRILLNG/MLLNG**
G265	**HEL. THREAD DRLG/MLG**
G267	**OUTSIDE THREAD MLLNG**
Cycles for milling pockets, studs and slots	
G233	**FACE MILLING**
G251	**RECTANGULAR POCKET**
G252	**CIRCULAR POCKET**
G253	**SLOT MILLING**
G254	**CIRCULAR SLOT**
G256	**RECTANGULAR STUD**
G257	**CIRCULAR STUD**
G258	**POLYGON STUD**
Cycles for creating point patterns	
G220	**POLAR PATTERN**
G221	**CARTESIAN PATTERN**
SL Cycles	
G37	**CONTOUR**
G120	**CONTOUR DATA** for G121 to G124
G121	**PILOT DRILLING**
G122	**ROUGH-OUT**
G123	**FLOOR FINISHING**
G124	**SIDE FINISHING**
G125	**CONTOUR TRAIN** for open contour
G270	**CONTOUR TRAIN DATA**
G127	**CYLINDER SURFACE**
G128	**CYLINDER SURFACE**
G129	**CYL SURFACE RIDGE**
G139	**CYL. SURFACE CONTOUR**
G275	**TROCHOIDAL SLOT**
G276	**THREE-D CONT. TRAIN**
Coordinate conversions	
G53	**DATUM SHIFT** from datum tables
G54	**DATUM SHIFT** in the program
G28	**MIRROR IMAGE**
G73	**ROTATION**
G72	**SCALING**
G80	**WORKING PLANE**
G247	**PRESETTING**
Cycles for multipass milling	
G230	**MULTIPASS MILLING**
G231	**RULED SURFACE**
*) blockwise effective function	

G codes	
Touch probe cycles for measuring workpiece misalignment	
G400	BASIC ROTATION
G401	ROT OF 2 HOLES
G402	ROT OF 2 STUDS
G403	ROT IN ROTARY AXIS
G404	SET BASIC ROTATION
G405	ROT IN C AXIS
Touch probe system cycles for setting datum	
G408	SLOT CENTER PRESET
G409	RIDGE CENTER PRESET
G410	PRESET INSIDE RECTAN
G411	PRESET OUTS. RECTAN
G412	PRESET INSIDE CIRCLE
G413	PRESET OUTS. CIRCLE
G414	PRESET OUTS. CORNER
G415	PRESET INSIDE CORNER
G416	PRESET CIRCLE CENTER
G417	PRESET IN TS AXIS
G418	PRESET FROM 4 HOLES
G419	PRESET IN ONE AXIS
Touch probe cycles for workpiece measurement	
G55	REF. PLANE
G420	MEASURE ANGLE
G421	MEASURE HOLE
G422	MEAS. CIRCLE OUTSIDE
G423	MEAS. RECTAN. INSIDE
G424	MEAS. RECTAN. OUTS.
G425	MEASURE INSIDE WIDTH
G426	MEASURE RIDGE WIDTH
G427	MEASURE COORDINATE
G430	MEAS. BOLT HOLE CIRC
G431	MEASURE PLANE
Touch probe cycles for tool measurement	
G480	CALIBRATE TT
G481	CAL. TOOL LENGTH
G482	CAL. TOOL RADIUS
G483	MEASURE TOOL
G434	CALIBRATE IR TT
Special cycles	
G04*	DWELL TIME
G36	ORIENTATION
G39*	PGM CALL
G62	TOLERANCE
Define the working plane	
G17	Spindle axis Z - plane XY
G18	Spindle axis Y - plane ZX
G19	Spindle axis X - plane YZ

G codes	
Dimensions	
G90	**Absolute dimension**
G91	**Incremental dimension**
Unit of measure	
G70	Unit of measure inch (at start of program)
G71	Unit of measure mm (at start of program)
Other G codes	
G29	**Load current position** (e.g. circle center as pole)
G38	**Stop program run**
G51*	**Prepare tool changer** (with central tool magazine)
G79*	**Cycle call**
G98*	**Set label**

*) blockwise effective function

Addresses	
%	Start of program
%	Program call
#	Datum number with G53
A	Rotation around the X axis
B	Rotation around the Y axis
C	Rotation around the Z axis
D	Q parameter definitions
DL	Wear compensation length with T
DR	Wear compensation radius with T
E	Tolerance with M112 and M124
F	Feed rate
F	Dwell time with G04
F	Scaling factor with G72
F	Factor F reduction with M103
G	G codes
H	Polar angle
H	Rotation angle with G73
H	Limit angle with M112
I	X coordinate of the circle center/pole
J	Y coordinate of the circle center/pole
K	Z coordinate of the circle center/pole
L	Setting a label number with G98
L	Jumping to a label number
L	Tool length with G99
M	M functions
N	Block number
P	Cycle parameter in machining cycles
P	Value or Q parameter in Q-parameter definition
Q	Q parameter

Addresses	
R R R R	Polar coordinate radius Circle radius with G02/G03/G05 Rounding radius with G25/G26/G27 Tool radius with G99
S S	Spindle speed Spindle orientation with G36
T T T	Tool definition with G99 Tool call Next tool with G51
U V W	Axis parallel to X axis Axis parallel to Y axis Axis parallel to Z axis
X Y Z	X axis Y axis Z axis
*	End of block

Contour cycles

Program structure with machining with multiple tools	
List of contour programs	G37 P01 ...
Defining **contour data**	G120 Q1 ...
Drill define/call Contour cycle: Pilot drilling Cycle call	G121 Q10 ...
Roughing mill define/call Contour cycle: Rough-out Cycle call	G122 Q10 ...
Finishing mill define/call Contour cycle: Floor finishing Cycle call	G123 Q11 ...
Finishing mill define/call Contour cycle: Side finishing Cycle call	G124 Q11 ...
End of main program, return	**M02**
Contour subprograms	G98 ... G98 L0

Radius compensation of the contour subprograms

Contour	Programming sequence of the contour elements	Radius Compensation
Inside (pocket)	clockwise (CW) counterclockwise (CCW)	G42 (RR) G41 (RL)
Outside (island)	clockwise (CW) counterclockwise (CCW)	G41 (RL) G42 (RR)

Coordinate conversions

Coordinate conversion	Activate	Cancel
Zero point shift	G54 X+20 Y+30 Z+10	G54 X0 Y0 Z0
Mirroring	G28 X	G28
Rotation	G73 H+45	G73 H+0
Scaling factor	G72 F 0.8	G72 F1 ...
Machining plane	G80 A+10 B+10 C+15	G80
Machining plane	PLANE ...	PLANE RESET

Q parameter definitions

D	Function
00	**Q parameter: Assign**
01	**Q parameter: Addition**
02	**Q parameter: Subtraction**
03	**Q parameter: Multiplication**
04	**Q parameter: Division**
05	**Q parameter: Square root**
06	**Q parameter: Sine**
07	**Q parameter: Cosine**
08	**Q parameter: Root sum of squares** $c = \sqrt{(a^2+b^2)}$
09	**Q parameter: If equal, go to** label number
10	**Q parameter: If unequal, go to** label number
11	**Q parameter: If greater, go to** label number
12	**Q parameter: If less than, go to** label number
13	**Q parameter: Angle with ARCTAN** (angle from c sin a and c cos a)
14	**Q parameter: Error message**
15	**Q parameter: External output**
16	**Q parameter: Write file**
18	**Q parameter: Read system data**
19	**Q parameter: Send value to PLC**

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

www.klartext-portal.com
The Information Site for HEIDENHAIN Controls

Klartext App
The Klartext on Your Mobile Device

Google Play Store

Apple App Store

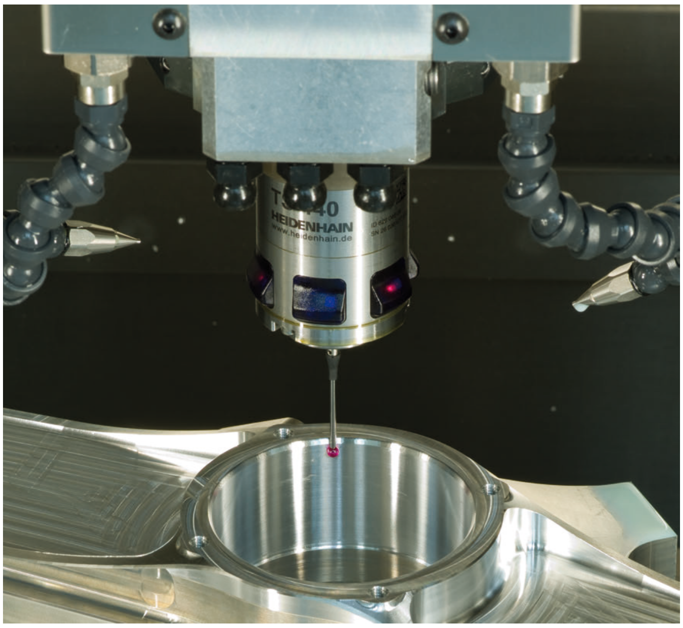
Touch probes from HEIDENHAIN

help you reduce non-productive time and improve the dimensional accuracy of the finished workpieces.

Workpiece touch probes

TS 220	Signal transmission by cable
TS 440, TS 444	Infrared transmission
TS 640, TS 740	Infrared transmission

- Workpiece alignment
- Setting presets
- Workpiece measurement



Tool touch probes

TT 140	Signal transmission by cable
TT 449	Infrared transmission
TL	Non-contacting laser systems

- Tool measurement
- wear monitoring
- tool breakage detection

Original document
892909-27 · Ver07 · SW09 · 10/2018 · H · Printed in Germany