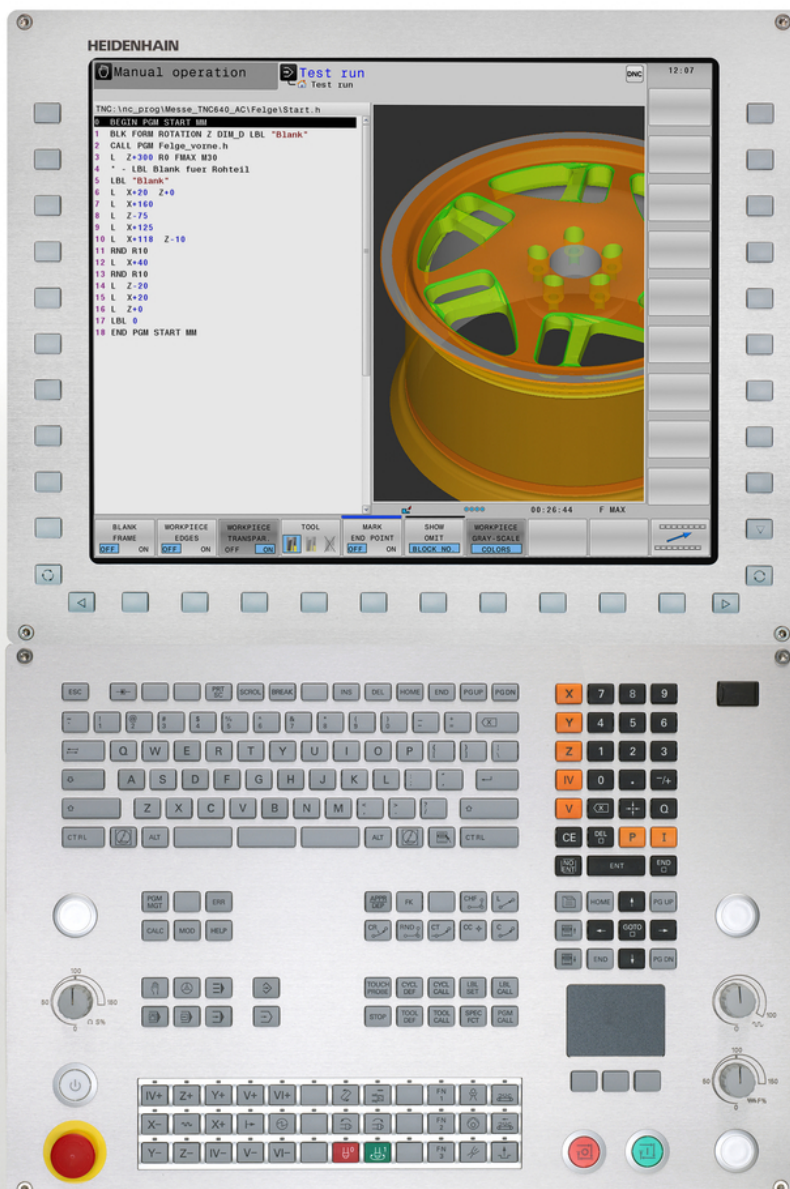




HEIDENHAIN



TNC 640

User's Manual
HEIDENHAIN
Conversational Programming

NC Software

340590-05

340591-05







340595-05

English (en)







1/2015

Controls of the TNC






Keys on visual display unit

Key	Function
	Select split screen layout
	Toggle the display between machining and programming modes
	Soft keys for selecting functions on screen
  	Shifting between soft-key rows



Alphanumeric keyboard

Key	Function
  	File names, comments
  	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



Program/file management, TNC functions

Key	Function
	Select or delete programs and files, external data transfer
	Define program call, select datum and point tables
	Select MOD functions
	Display help text for NC error messages, call TNCguide
	Display all current error messages
	Show calculator







Navigation keys

Key	Function
 	Move highlight
	Go directly to blocks, cycles and parameter functions



Potentiometer for feed rate and spindle speed

Feed rate	Spindle speed
	










Cycles, subprograms and program section repeats

Key	Function
	Define touch probe cycles
 	Define and call cycles
 	Enter and call labels for subprogramming and program section repeats
	Enter program stop in a program





Tool functions

Key	Function
	Define tool data in the program
	Call tool data
















Programming path movements

Key	Function
	Approach/depart contour
	FK free contour programming
	Straight line
	Circle center/pole for polar coordinates
	Circular arc with center
	Circle with radius
	Circular arc with tangential connection
 	Chamfer/Corner rounding

Special functions

Key	Function
	Show special functions
	Select the next tab in forms
 	Up/down one dialog box or button

Entering and editing coordinate axes and numbers

Key	Function
 ... 	Select coordinate axes or enter them in a program
 ... 	Numbers
 	Decimal point / Reverse algebraic sign
 	Polar coordinate input / Incremental values
	Q-parameter programming / Q-parameter status
	Save actual position or values from calculator
	Skip dialog questions, delete words
	Confirm entry and resume dialog
	Conclude block and exit entry
	Clear numerical entry or TNC error message
	Abort dialog, delete program section

Fundamentals

About this manual

The symbols used in this manual are described below.



This symbol indicates that important information about the function described must be considered.



This symbol indicates that there is one or more of the following risks when using the described function:

- Danger to workpiece
- Danger to fixtures
- Danger to tool
- Danger to machine
- Danger to operator



This symbol indicates a possibly dangerous situation that may cause injuries if not avoided.



This symbol indicates that the described function must be adapted by the machine tool builder. The function described may therefore vary depending on the machine.



This symbol indicates that you can find detailed information about a function in another manual.

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

We are continuously striving to improve our documentation for you. Please help us by sending your requests to the following e-mail address: **tnc-userdoc@heidenhain.de**.

TNC model, software and features

This manual describes functions and features provided by TNCs as of the following NC software numbers.

TNC model	NC software number
TNC 640	340590-05
TNC 640 E	340591-05
TNC 640 Programming Station	340595-05

The suffix E indicates the export version of the TNC. The export version of the TNC has the following limitations:

- Simultaneous linear movement in up to 4 axes

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

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

- Tool measurement with the TT

Please contact your machine tool builder to become familiar with the features of your machine.

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



User's Manual for Cycle Programming:

All of the cycle functions (touch probe cycles and fixed cycles) are described in the Cycle Programming User's Manual. Please contact HEIDENHAIN if you require a copy of this User's Manual. ID: 892905-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:

Hardware, options

- 1st additional axis for 4 axes plus spindle
- 2nd additional axis for 5 axes plus spindle

Software option 1 (option number 08)

- | | | |
|-------------------------------|---|---|
| Rotary table machining | ■ | Programming of cylindrical contours as if in two axes |
| | ■ | Feed rate in distance per minute |
-

- | | | |
|----------------------------------|---|--------------------------------|
| Coordinate transformation | ■ | Working plane, tilting the ... |
|----------------------------------|---|--------------------------------|
-

- | | | |
|----------------------|---|--|
| Interpolation | ■ | Circle in 3 axes with tilted working plane (spacial arc) |
|----------------------|---|--|
-

Software option 2 (option number 09)

- | | | |
|----------------------|---|--|
| 3-D machining | ■ | Motion control with minimum jerk |
| | ■ | 3-D tool compensation through surface normal vectors |
| | ■ | Using the electronic handwheel to change the angle of the swivel head during program run without affecting the position of the tool point. (TCPM = T ool C enter P oint M anagement) |
| | ■ | Keeping the tool normal to the contour |
| | ■ | Tool radius compensation perpendicular to traversing and tool direction |
-

- | | | |
|----------------------|---|---|
| Interpolation | ■ | Linear in 5 axes (subject to export permit) |
|----------------------|---|---|
-

HEIDENHAIN DNC (option number 18)

- Communication with external PC applications over COM component

Display step (Option number 23)

- | | | |
|--|---|-------------------------|
| Input resolution and display step | ■ | Linear axes to 0.01 µm |
| | ■ | Rotary axes to 0.00001° |
-

Dynamic Collision Monitoring (DCM) software option (option number 40)

- | | | |
|--|---|--|
| Collision monitoring in all machine operating modes | ■ | The machine manufacturer defines objects to be monitored |
| | ■ | Three warning levels in manual operation |
| | ■ | Program interrupt during automatic operation |
| | ■ | Includes monitoring of 5-axis movements |
-

Software option for additional conversational languages (option number 41)

- | | | |
|--|---|-----------|
| Additional conversational languages | ■ | Slovenian |
| | ■ | Norwegian |
| | ■ | Slovak |
| | ■ | Latvian |
| | ■ | Korean |
| | ■ | Estonian |
| | ■ | Turkish |
| | ■ | Romanian |

Software option for additional conversational languages (option number 41)

- Lithuanian

DXF Converter software option (option number 42)

- | | |
|---|---|
| Extracting contour programs and machining positions from DXF data. Extracting contour sections from plain-language programs. | <ul style="list-style-type: none">■ Supported DXF format: AC1009 (AutoCAD R12)■ For contours and point patterns■ Simple and convenient specification of reference points■ Select graphical features of contour sections from conversational programs |
|---|---|

Adaptive Feed Control (AFC) software option (option number 45)

- | | |
|---|--|
| Function for adaptive feed-rate control for optimizing the machining conditions during series production | <ul style="list-style-type: none">■ 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 |
|---|--|

KinematicsOpt software option (option number 48)

- | | |
|--|--|
| Touch-probe cycles for automatic testing and optimization of the machine kinematics | <ul style="list-style-type: none">■ Backup/restore active kinematics■ Test active kinematics■ Optimize active kinematics |
|--|--|

Mill-Turning software option (option number 50)

- | | |
|---|--|
| Functions for milling/turning mode | <ul style="list-style-type: none">■ Switching between Milling/Turning mode of operation■ Constant cutting speed■ Tool-tip radius compensation■ Turning cycles |
|---|--|

Extended Tool Managment software option (option number 93)

- Extended tool management, python-based

Remote Desktop Manager software option (option number 133)

- | | |
|---|---|
| Remote operation of external computer units (e.g. Windows PC) via the TNC user interface | <ul style="list-style-type: none">■ Windows on a separate computer unit■ Incorporated in the TNC interface |
|---|---|

Cross Talk Compensation (CTC) software option (option number 141)

- | | |
|---------------------------------------|--|
| Compensation of axis couplings | <ul style="list-style-type: none">■ Determination of dynamically caused position deviation through axis acceleration■ Compensation of the TCP |
|---------------------------------------|--|

Position Adaptive Control (PAC) software option (option number 142)

- | | |
|------------------------------------|--|
| Changing control parameters | <ul style="list-style-type: none">■ 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) software option (option number 143)

- | | |
|---|---|
| Dynamic changing of control parameters | <ul style="list-style-type: none">■ Automatic determination of workpiece weight and frictional forces■ Continuous adaptation of the parameters of the adaptive precontrolling to the actual weight of the workpiece during machining |
|---|---|

Active Chatter Control (ACC) software option (option number 145)

Fully automatic function for chatter control during machining

Feature Content Level (upgrade functions)

Along with software options, significant further improvements of the TNC software are managed via the **Feature Content Level** upgrade functions. Functions subject to the FCL are not available simply by updating the software on your TNC.



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

Upgrade functions are identified in the manual with **FCL n**, where **n** indicates the sequential number of the feature content level.

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 TNC 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 under

- ▶ Programming and Editing operating mode
- ▶ MOD function
- ▶ **LICENSE INFO** soft key

New functions

New functions 34059x-02

DXF files can be opened directly on the TNC in order to extract contours and point patterns ("Programming: Data Transfer from CAD Files", page 257).

The active tool-axis direction can now be activated in manual mode and during handwheel superimposition as a virtual tool axis ("Superimposing handwheel positioning during program run: M118", page 380).

The machine manufacturer can now define any areas on the machine for collision monitoring ("Dynamic Collision Monitoring (Option #40)", page 393).

Writing and reading data in freely definable tables ("Freely definable tables", page 427).

The Adaptive Feed Control (AFC) function has been integrated ("Adaptive feed control AFC (Option #45)", page 400)

New touch probe cycle 484 for calibrating the wireless TT 449 tool touch probe (see User's Manual for Cycles).

The new HR 520 and HR 550 FS handwheels are supported ("Traverse with electronic handwheels", page 520).

New machining cycle 225 ENGRAVING (see User's Manual for Cycle Programming)

New Active Chatter Control (ACC) software option ("Active Chatter Control ACC (Option #145)", page 412).

New manual probing cycle "Center line as datum" ("Setting a center line as datum", page 567).

New function for rounding corners ("Rounding corners: M197", page 387).

External access to the TNC can now be blocked with a MOD function ("External access", page 615).

Changed functions 34059x-02

The maximum number of characters for the NAME and DOC fields in the tool table has been increased from 16 to 32 ("Enter tool data into the table", page 174).

The columns AFC and ACC were added to the tool table ("Enter tool data into the table", page 174).

Operation and positioning behavior of the manual probing cycles has been improved ("Using 3-D touch probes ", page 545).

Predefined values can now be entered into a cycle parameter with the PREDEF function in cycles (see User's Manual for Cycle Programming).

The status display has been expanded with the AFC tab ("Additional status displays", page 79).

The FUNCTION TURNDATA SPIN rotational function has been expanded with an input option for maximum speed ("Program spindle speed", page 492).

A new optimization algorithm is now used with the KinematicsOpt cycles (see User's Manual for Cycle Programming).

With Cycle 257, circular stud milling, a parameter is now available with which you can determine the approach position on the stud (see User's Manual for Cycle Programming)

With Cycle 256, rectangular stud, a parameter is now available with which you can determine the approach position on the stud (see User's Manual for Cycle Programming).

With the "Basic Rotation" probing cycle, workpiece misalignment can now be compensated for via a table rotation ("Compensation of workpiece misalignment by rotating the table", page 560)

New functions 34059x-04

New special operating mode ("Retraction after a power interruption", page 603).

New graphic simulation ("Graphics ", page 584).

New MOD function "tool usage file" within the machine settings group ("Tool usage file", page 618).

New MOD function "set system time" within the systems settings group ("Set the system time", page 620).

New MOD group "graphic settings" ("Graphic settings", page 614).

With the new syntax for the adaptive feed control (AFC) you can start or end a teach-in step ("Recording a teach-in cut", page 404).

With the new cutting data calculator you can calculate the spindle speed and the feed rate ("Cutting data calculator", page 149).

In the TURNDATA function, you can now define the effect of the tool compensation ("Tool compensation in the program", page 498).

Now you can activate and deactivate the active chatter compensation (ACC) by soft key ("Activating/deactivating ACC", page 413).

New if/then decisions were introduced in the jump commands ("Programming if-then decisions", page 308).

The character set of the fixed cycle 225 Engraving was expanded by more characters and the diameter sign (see User's Manual for Cycle Programming).

New fixed cycle 275 Trochoidal Milling (see User's Manual for Cycle Programming)

New fixed cycle 233 ENGRAVING (see User's Manual for Cycle Programming)

In the drilling cycles 200, 203 and 205 the parameter Q395 DEPTH REFERENCE was introduced in order to evaluate the T ANGLE (see User's Manual for Cycle Programming).

The probing cycle 4 MEASURING IN 3-D was introduced (see User's Manual for Cycle Programming).

Changed functions 34059x-04

The turning tool table was expanded by the column NAME ("Tool data", page 499).

Now up to 4 functions are allowed in an NC block ("Fundamentals", page 368).

New soft keys for value transfer have been introduced in the pocket calculator ("Operation", page 146).

The distance-to-go display can now also be displayed in the input system ("Select the position display", page 621).

Cycle 241 SINGLE-LIP DEEP HOLE DRILLING was expanded by several input parameters (see User's Manual for Cycle Programming).

Cycle 404 was expanded by the parameter Q305 NUMBER IN TABLE (see User's Manual for Cycle Programming).

In the thread milling cycles 26x an approaching feed rate was introduced (see User's Manual for Cycle Programming).

In Cycle 205 Universal Pecking you can now use parameter Q208 to define a feed rate for retraction (see User's Manual for Cycle Programming).

New functions 34059x-05

The tool management was expanded by the column PITCH ("Enter tool data into the table", page 174).

The turning tool table was expanded by the columns YL and DYL ("Tool data", page 499).

In the tool management, several lines can now be added at the end of the table ("Tool management (Option #93)", page 192).

Any turning tool table can be selected for the program test ("Test Run", page 596).

Programs with .HU and .HC endings can be selected and processed in all operating modes.

The functions `and` `have` been added ("Calling any program as a subprogram").

New **FEED DWELL** function for programming repeating dwell times ("Dwell time FUNCTION FEED DWELL").

The FN 18 functions have been expanded ("FN 18: SYSREAD: Reading system data", page 320).

The DCM function can be activated and deactivated from the NC program ("Activating and deactivating collision monitoring", page 398).

USB data carriers can be locked with the SELinux security software ("SELinux security software", page 92).

The `posAfterContPocket` machine parameter has been added that influences positioning after an SL cycle ("Machine-specific user parameters", page 646).

Protective zones can be defined in the MOD menu ("Entering traverse limits", page 617).

Write protection is possible for single lines in the preset table ("Saving the datums in the preset table", page 537).

New manual probing function for aligning a plane ("Measuring 3-D basic rotation", page 561).

New function for aligning the machining plane without rotary axes ("Tilt the working plane without rotary axes", page 458).

CAD files can be opened without Option #42 ("CAD Viewer", page 259).

New software option #131 Spindle Synchronism ("Software options", page 8).

Modified functions 34059x-05

FZ and FU feed rate input possible in the Tool Call block ("Call tool data", page 186).

With tool selection, the control also displays columns XL and ZL from the turning tool table in the pop-up window ("Tool call", page 497).

The input range of the DOC column in the pocket table has been expanded to 32 characters ("Pocket table for tool changer").

Commands FN 15, FN 31, FN 32, FT and FMAXT from predecessor controls no longer generate ERROR blocks during import. When simulating or running an NC program with these commands, the control interrupts the NC program with an error message that helps you to find an alternative implementation.

Miscellaneous functions M104, M105, M112, M114, M124, M134, M142, M150, M200 - M204 from predecessor controls no longer generate ERROR blocks during import. When simulating or running an NC program with these miscellaneous functions, the control interrupts the NC program with an error message that helps you to find an alternative implementation ("Comparison: Miscellaneous functions").

The maximum file size of files output with FN 16: F-PRINT has been increased from 4kB to 20kB.

The Preset.PR preset table is write-protected in Programming operating mode ("Saving the datums in the preset table").

The input range of the Q parameter list for defining the QPARA tab on the status display consists of 132 input positions ("Displaying Q parameters (QPARA tab)", page 84).

Manual calibration of the touch probe with less pre-positionings ("Calibrating a 3-D touch trigger probe").

The position display takes into account the DL oversizes programmed in the Tool Call block, selectable as an oversize of the workpiece or tool ("Delta values for lengths and radii", page 173).

In single blocks, the control executes each point singly with point pattern cycles and CYCL CALL PAT ("Program run", page 598).

Rebooting the control is no longer possible with the **END** key, but with the soft key ("Switch-off", page 518).

The control displays the contouring feed rate in manual mode ("Spindle speed S, feed rate F and miscellaneous function M", page 530).

Deactivate tilting in manual mode is only possible via the 3D-ROT menu ("To activate manual tilting:", page 574).

The machine parameter maxLineGeoSearch has been increased to a maximum of 100000 ("Machine-specific user parameters", page 646).

The names of the software options #8, #9 and #21 have been changed ("Software options", page 8).

New and modified cycle functions 34059x-05

New cycle **880** (Option #50), see "ZAHNRAD ABWÄLZFRÄSEN (Zyklus 880, DIN/ISO: G880)"

New cycle **292** (Option #96), see "INTERPOLATIONS DREHEN KONTURSCHLICHTEN (Zyklus 292, DIN/ISO: G292, Softwareoption 96)"

New cycle **291 COUPLG. TURNG. INTERP.** (Option #96), see "INTERPOLATIONS DREHEN KOPPLUNG (Zyklus 291, DIN/ISO: G291, Softwareoption 96)"

New cycle **239** for LAC (Load Adapt. Control) load-dependent adaptation of control parameters (Option #143), see "BELADUNG ERMITTELN (Zyklus 239 DIN/ISO: G239, Software-Option 143)"

Cycle **270 CONTOUR TRAIN DATA** has been added (Option #19), see "KONTURZUG-DATEN (Zyklus 270, DIN/ISO: G270, Software-Option 19)"

Cycle **39** has been added (Option #1), see "ZYLINDER-MANTEL (Zyklus 39, DIN/ISO: G139, Software-Option 1)"

The character set of machining cycle **225** has been expanded with the CE character, ß, the @ character and system time, see "ENGRAVING (Cycle 225, DIN/ISO: G225)"

Cycles **252-254** have been expanded with the optional parameter Q439

Cycle **22** has been expanded with the optional parameters Q401, Q404, see "ROUGHING (Cycle 22, DIN/ISO: G122, software option 19)"

Cycle **484** has been expanded with the optional parameter Q536, see "Calibrate the wireless TT 449 (Cycle 484, DIN/ISO: G484, software option 17 Touch Probe Functions software option 17)"

Cycles **841 SIMPLE REC. TURNG., RADIAL DIR., 842 , 851 , 852** have been expanded with plunge feed rate Q488

Eccentric turning with cycle **800** is possible with Option #50, see "ADAPT ROTARY COORDINATE SYSTEM (Cycle 800, DIN/ISO: G800)"

Contents

1	First Steps with the TNC 640.....	51
2	Introduction.....	71
3	Programming: Fundamentals, File Management.....	95
4	Programming: Programming aids.....	141
5	Programming: Tools.....	169
6	Programming: Programming contours.....	205
7	Programming: Data Transfer from CAD Files.....	257
8	Programming: Subprograms and program section repeats.....	279
9	Programming: Q Parameters.....	297
10	Programming: Miscellaneous functions.....	367
11	Programming: Special functions.....	389
12	Programming: Multiple Axis Machining.....	435
13	Programming: Pallet editor.....	481
14	Programming: Turning Operations.....	487
15	Manual operation and setup.....	515
16	Positioning with Manual Data Input.....	577
17	Test run and program run.....	583
18	MOD functions.....	611
19	Tables and overviews.....	645

1	First Steps with the TNC 640.....	51
1.1	Overview.....	52
1.2	Machine switch-on.....	52
	Acknowledging the power interruption and moving to the reference points.....	52
1.3	Programming the first part.....	53
	Selecting the correct operating mode.....	53
	The most important TNC keys.....	53
	Opening a new program/file management.....	54
	Defining a workpiece blank.....	55
	Program layout.....	56
	Programming a simple contour.....	57
	Creating a cycle program.....	60
1.4	Graphically testing the first part.....	62
	Selecting the correct operating mode.....	62
	Selecting the tool table for the test run.....	62
	Choosing the program you want to test.....	63
	Selecting the screen layout and the view.....	63
	Starting the test run.....	64
1.5	Setting up tools.....	65
	Selecting the correct operating mode.....	65
	Preparing and measuring tools.....	65
	The tool table TOOL.T.....	66
	The pocket table TOOL_PTCH.....	67
1.6	Workpiece setup.....	68
	Selecting the correct operating mode.....	68
	Clamping the workpiece.....	68
	Datum setting with 3-D touch probe.....	69
1.7	Running the first program.....	70
	Selecting the correct operating mode.....	70
	Choosing the program you want to run.....	70
	Start the program.....	70

2	Introduction.....	71
2.1	The TNC 640.....	72
	Programming: In HEIDENHAIN conversational and DIN/ISO.....	72
	Compatibility.....	72
2.2	Visual display unit and operating panel.....	73
	Display screen.....	73
	Setting the screen layout.....	73
	Control Panel.....	74
2.3	Modes of Operation.....	75
	Manual Operation and El. Handwheel.....	75
	Positioning with Manual Data Input.....	75
	Programming.....	76
	Test Run.....	76
	Program Run, Full Sequence and Program Run, Single Block.....	77
2.4	Status displays.....	78
	General status display.....	78
	Additional status displays.....	79
2.5	Window Manager.....	86
	Task bar.....	87
2.6	Remote Desktop Manager (Option #133).....	88
	Introduction.....	88
	Configuring connections – Windows Terminal Service.....	88
	Configuring the connection – VNC.....	90
	Starting and stopping the connection.....	91
2.7	SELinux security software.....	92
2.8	Accessories: HEIDENHAIN 3-D Touch Probes and Electronic Handwheels.....	93
	3-D touch probes.....	93
	HR electronic handwheels.....	94

3 Programming: Fundamentals, File Management..... 95

3.1 Fundamentals..... 96

Position encoders and reference marks.....	96
Reference system.....	96
Reference system on milling machines.....	97
Designation of the axes on milling machines.....	97
Polar coordinates.....	98
Absolute and incremental workpiece positions.....	99
Selecting the datum.....	100

3.2 Opening programs and entering..... 101

Organization of an NC program in HEIDENHAIN Conversational format.....	101
Define the blank: BLK FORM.....	102
Opening a new part program.....	104
Programming tool movements in conversational.....	106
Actual position capture.....	108
Editing a program.....	109
The TNC search function.....	112

3.3 File Management: Fundamentals..... 114

Files.....	114
Displaying externally generated files on the TNC.....	116
Data Backup.....	116

3.4 Working with the File Manager.....	117
Directories.....	117
Paths.....	117
Overview: Functions of the file manager.....	118
Calling the File Manager.....	119
Selecting drives, directories and files.....	120
Creating a new directory.....	121
Creating a new file.....	121
Copying a single file.....	121
Copying files into another directory.....	122
Copying a table.....	123
Copying a directory.....	123
Choosing one of the last files selected.....	124
Deleting a file.....	125
Deleting a directory.....	125
Tagging files.....	126
Renaming a file.....	127
Sorting files.....	127
Additional functions.....	128
Additional tools for management of external file types.....	129
Data transfer to/from an external data medium.....	136
The TNC in a network.....	138
USB devices on the TNC.....	139

4	Programming: Programming aids.....	141
4.1	Adding comments.....	142
	Application.....	142
	Entering comments during programming.....	142
	Inserting comments after program entry.....	142
	Entering a comment in a separate block.....	142
	Functions for editing of the comment.....	143
4.2	Display of NC Programs.....	144
	Syntax highlighting.....	144
	Scrollbar.....	144
4.3	Structuring programs.....	145
	Definition and applications.....	145
	Displaying the program structure window / Changing the active window.....	145
	Inserting a structuring block in the program window.....	145
	Selecting blocks in the program structure window.....	145
4.4	Calculator.....	146
	Operation.....	146
4.5	Cutting data calculator.....	149
	Application.....	149
4.6	Programming graphics.....	152
	Generate/do not generate graphics during programming.....	152
	Generating a graphic for an existing program.....	153
	Block number display ON/OFF.....	154
	Erasing the graphic.....	154
	Showing grid lines.....	154
	Magnification or reduction of details.....	155

4.7 Error messages..... 156

Display of errors.....	156
Open the error window.....	156
Closing the error window.....	156
Detailed error messages.....	157
INTERNAL INFO soft key.....	157
Clearing errors.....	158
Error log.....	158
Keystroke log.....	159
Informational texts.....	160
Saving service files.....	161
Calling the TNCguide help system.....	161

4.8 TNCguide context-sensitive help system..... 162

Application.....	162
Working with the TNCguide.....	163
Downloading current help files.....	167

5 Programming: Tools..... 169

5.1 Entering tool-related data..... 170

Feed rate F..... 170

Spindle speed S..... 171

5.2 Tool data..... 172

Requirements for tool compensation..... 172

Tool number, tool name..... 172

Tool length L..... 172

Tool radius R..... 172

Delta values for lengths and radii..... 173

Entering tool data into the program..... 173

Enter tool data into the table..... 174

Importing tool tables..... 182

Pocket table for tool changer..... 183

Call tool data..... 186

Tool change..... 188

Tool usage test..... 190

Tool management (Option #93)..... 192

5.3 Tool compensation..... 200

Introduction..... 200

Tool length compensation..... 200

Tool radius compensation..... 201

6	Programming: Programming contours.....	205
6.1	Tool movements.....	206
	Path functions.....	206
	FK free contour programming.....	206
	Miscellaneous functions M.....	206
	Subprograms and program section repeats.....	207
	Programming with Q parameters.....	207
6.2	Fundamentals of Path Functions.....	208
	Programming tool movements for workpiece machining.....	208
6.3	Approaching and departing a contour.....	212
	Starting point and end point.....	212
	Overview: Types of paths for contour approach and departure.....	214
	Important positions for approach and departure.....	215
	Approaching on a straight line with tangential connection: APPR LT.....	217
	Approaching on a straight line perpendicular to the first contour point: APPR LN.....	217
	Approaching on a circular path with tangential connection: APPR CT.....	218
	Approaching on a circular path with tangential connection from a straight line to the contour: APPR LCT.....	219
	Departing in a straight line with tangential connection: DEP LT.....	219
	Departing in a straight line perpendicular to the last contour point: DEP LN.....	220
	Departing on a circular path with tangential connection: DEP CT.....	221
	Departing on a circular arc tangentially connecting the contour and a straight line: DEP LCT.....	221
6.4	Path contours - Cartesian coordinates.....	222
	Overview of path functions.....	222
	Straight line L.....	223
	Inserting a chamfer between two straight lines.....	224
	Corner rounding RND.....	225
	Circle center CC.....	226
	Circular path C around circle center CC.....	227
	CircleCR with defined radius.....	228
	Circle CT with tangential connection.....	230
	Example: Linear movements and chamfers with Cartesian coordinates.....	231
	Example: Circular movements with Cartesian coordinates.....	232
	Example: Full circle with Cartesian coordinates.....	233

6.5 Path contours – Polar coordinates..... 234

Overview.....	234
Zero point for polar coordinates: pole CC.....	235
Straight line LP.....	235
Circular path CP around pole CC.....	236
Circle CTP with tangential connection.....	236
Helix.....	237
Example: Linear movement with polar coordinates.....	239
Example: Helix.....	240

6.6 Path contours – FK free contour programming..... 241

Fundamentals.....	241
FK programming graphics.....	243
Initiating the FK dialog.....	244
Pole for FK programming.....	244
Free straight line programming.....	245
Free circular path programming.....	246
Input options.....	247
Auxiliary points.....	250
Relative data.....	251
Example: FK programming 1.....	253
Example: FK programming 2.....	254
Example: FK programming 3.....	255

7 Programming: Data Transfer from CAD Files..... 257

7.1 CAD viewer and DXF converter screen layout.....258

CAD viewer and DXF converter screen layout..... 258

7.2 CAD Viewer.....259

Application..... 259

7.3 DXF converter (Option #42)..... 260

Application..... 260

Working with the DXF converter..... 261

Opening a DXF file..... 262

Basic settings..... 263

Setting layers..... 265

Defining the datum..... 266

Selecting and saving a contour..... 268

Selecting and saving machining positions..... 272

8	Programming: Subprograms and program section repeats.....	279
8.1	Labeling Subprograms and Program Section Repeats.....	280
	Label.....	280
8.2	Subprograms.....	281
	Operating sequence.....	281
	Programming notes.....	281
	Programming a subprogram.....	281
	Calling a subprogram.....	282
8.3	Program-section repeats.....	283
	Label.....	283
	Operating sequence.....	283
	Programming notes.....	283
	Programming a program section repeat.....	283
	Calling a program section repeat.....	284
8.4	Any desired program as subprogram.....	285
	Overview of the soft keys.....	285
	Operating sequence.....	286
	Programming notes.....	286
	Calling any program as a subprogram.....	287
8.5	Nesting.....	289
	Types of nesting.....	289
	Nesting depth.....	289
	Subprogram within a subprogram.....	290
	Repeating program section repeats.....	291
	Repeating a subprogram.....	292
8.6	Programming examples.....	293
	Example: Milling a contour in several infeeds.....	293
	Example: Groups of holes.....	294
	Example: Group of holes with several tools.....	295

9	Programming: Q Parameters.....	297
9.1	Principle and overview of functions.....	298
	Programming notes.....	300
	Calling Q parameter functions.....	301
9.2	Part families—Q parameters in place of numerical values.....	302
	Application.....	302
9.3	Describing contours with mathematical functions.....	303
	Application.....	303
	Overview.....	303
	Programming fundamental operations.....	304
9.4	Angle functions.....	305
	Definitions.....	305
	Programming trigonometric functions.....	305
9.5	Calculation of circles.....	306
	Application.....	306
9.6	If-then decisions with Q parameters.....	307
	Application.....	307
	Unconditional jumps.....	307
	Abbreviations used:.....	307
	Programming if-then decisions.....	308
9.7	Checking and changing Q parameters.....	309
	Procedure.....	309
9.8	Additional functions.....	311
	Overview.....	311
	FN 14: ERROR: Displaying error messages.....	312
	FN16: F-PRINT – Formatted output of text and Q parameter values.....	316
	FN 18: SYSREAD: Reading system data.....	320
	FN 19: PLC – Transfer values to the PLC.....	329
	FN 20: WAIT FOR – NC and PLC synchronization.....	329
	FN 29: PLC – Transfer values to the PLC.....	330
	FN 37: EXPORT.....	330

9.9 Accessing tables with SQL commands..... 331

Introduction.....	331
A transaction.....	332
Programming SQL commands.....	334
Overview of the soft keys.....	334
SQL BIND.....	335
SQL SELECT.....	336
SQL FETCH.....	338
SQL UPDATE.....	339
SQL INSERT.....	339
SQL COMMIT.....	340
SQL ROLLBACK.....	340

9.10 Entering formulas directly..... 341

Entering formulas.....	341
Rules for formulas.....	343
Programming example.....	344

9.11 String parameters..... 345

String processing functions.....	345
Assigning string parameters.....	346
Chain-linking string parameters.....	346
Converting a numerical value to a string parameter.....	347
Copying a substring from a string parameter.....	348
Converting a string parameter to a numerical value.....	349
Checking a string parameter.....	350
Finding the length of a string parameter.....	351
Comparing alphabetic sequence.....	352
Reading out machine parameters.....	353

9.12 Preassigned Q parameters..... 356

Values from the PLC: Q100 to Q107.....	356
Active tool radius: Q108.....	356
Tool axis: Q109.....	356
Spindle status: Q110.....	357
Coolant on/off: Q111.....	357
Overlap factor: Q112.....	357
Unit of measurement for dimensions in the program: Q113.....	357
Tool length: Q114.....	357
Coordinates after probing during program run.....	358
Deviation between actual value and nominal value during automatic tool measurement with the TT 130.....	358
Tilting the working plane with mathematical angles: rotary axis coordinates calculated by the TNC.....	358
Measurement results from touch probe cycles (see also User's Manual for Cycle Programming).....	359

9.13 Programming examples..... 361

Example: Ellipse.....	361
Example: Concave cylinder machined with spherical cutter.....	363
Example: Convex sphere machined with end mill.....	365

10 Programming: Miscellaneous functions.....	367
10.1 Entering miscellaneous functions M and STOP.....	368
Fundamentals.....	368
10.2 M functions for program run inspection, spindle and coolant.....	369
Overview.....	369
10.3 Miscellaneous functions for coordinate data.....	370
Programming machine-referenced coordinates: M91/M92.....	370
Moving to positions in a non-tilted coordinate system with a tilted working plane: M130.....	372
10.4 Miscellaneous functions for path behavior.....	373
Machining small contour steps: M97.....	373
Machining open contour corners: M98.....	374
Feed rate factor for plunging movements: M103.....	375
Feed rate in millimeters per spindle revolution: M136.....	376
Feed rate for circular arcs: M109/M110/M111.....	377
Calculating the radius-compensated path in advance (LOOK AHEAD): M120.....	378
Superimposing handwheel positioning during program run: M118.....	380
Retraction from the contour in the tool-axis direction: M140.....	382
Suppressing touch probe monitoring: M141.....	384
Deleting basic rotation: M143.....	385
Automatically retract tool from the contour at an NC stop: M148.....	386
Rounding corners: M197.....	387

11 Programming: Special functions.....	389
11.1 Overview of special functions.....	390
Main menu for SPEC FCT special functions.....	390
Program defaults menu.....	391
Functions for contour and point machining menu.....	391
Menu of various conversational functions.....	392
11.2 Dynamic Collision Monitoring (Option #40).....	393
Function.....	393
Graphic display of the collision objects.....	394
Collision monitoring in the manual operating modes.....	396
Collision monitoring in the Program Run operating modes.....	396
Activating and deactivating collision monitoring.....	398
11.3 Adaptive feed control AFC (Option #45).....	400
Application.....	400
Defining the AFC basic settings.....	402
Recording a teach-in cut.....	404
Activating/deactivating AFC.....	407
Log file.....	409
Tool breakage/tool wear monitoring.....	410
Spindle load monitoring.....	411
11.4 Active Chatter Control ACC (Option #145).....	412
Application.....	412
Activating/deactivating ACC.....	413
11.5 Working with the Parallel Axes U, V and W.....	414
Overview.....	414
FUNCTION PARAXCOMP DISPLAY.....	415
FUNCTION PARAXCOMP MOVE.....	415
Deactivating FUNCTION PARAXCOMP.....	416
FUNCTION PARAXMODE.....	416
Deactivating FUNCTION PARAXMODE.....	417
Example: Drilling in the W axis.....	418
11.6 File functions.....	419
Application.....	419
Defining file functions.....	419

11.7 Definition of a datum shift..... 420

Overview.....	420
TRANS DATUM AXIS.....	420
TRANS DATUM TABLE.....	421
TRANS DATUM RESET.....	422

11.8 Creating Text Files.....423

Application.....	423
Opening and exiting text files.....	423
Editing texts.....	424
Deleting and re-inserting characters, words and lines.....	424
Editing text blocks.....	425
Finding text sections.....	426

11.9 Freely definable tables..... 427

Fundamentals.....	427
Creating a freely definable table.....	427
Editing the table format.....	428
Switching between table and form view.....	429
FN 26: TABOPEN – Open a freely definable table.....	430
FN 27: TABWRITE – Write to a freely definable table.....	431
FN 28: TABREAD – Read from a freely definable table.....	432

11.10 Dwell time FUNCTION FEED DWELL.....433

Programming dwell time.....	433
Resetting dwell time.....	434

12 Programming: Multiple Axis Machining.....	435
12.1 Functions for multiple axis machining.....	436
12.2 The PLANE Function: Tilting the working plane (Software Option 8).....	437
Introduction.....	437
Overview.....	438
Defining the PLANE function.....	439
Position display.....	439
Resetting the PLANE function.....	440
Defining the working plane with the spatial angle: PLANE SPATIAL.....	441
Defining the working plane with the projection angle: PLANE PROJECTED.....	443
Defining the working plane with the Euler angle: PLANE EULER.....	444
Defining the working plane with two vectors: PLANE VECTOR.....	446
Defining the working plane via three points: PLANE POINTS.....	448
Defining the working plane via a single incremental spatial angle: PLANE SPATIAL.....	450
Tilting the working plane through axis angle: PLANE AXIAL (FCL 3 function).....	451
Specifying the positioning behavior of the PLANE function.....	453
Tilt the working plane without rotary axes.....	458
12.3 Inclined-tool machining in a tilted plane (Option #9).....	459
Function.....	459
Inclined-tool machining via incremental traverse of a rotary axis.....	459
Inclined-tool machining via normal vectors.....	460
12.4 Miscellaneous functions for rotary axes.....	461
Feed rate in mm/min on rotary axes A, B, C: M116 (Option #8).....	461
Shortest-path traverse of rotary axes: M126.....	462
Reducing display of a rotary axis to a value less than 360°: M94.....	463
Maintaining the position of the tool tip when positioning with tilted axes (TCPM): M128 (Option #9).....	464
Selecting tilting axes: M138.....	467
Compensating the machine's kinematics configuration for ACTUAL/NOMINAL positions at end of block: M144 (Option #9).....	468

12.5 FUNCTION TCPM (Option #9).....469

Function.....	469
Defining the TCPM FUNCTION.....	469
Mode of action of the programmed feed rate.....	470
Interpretation of the programmed rotary axis coordinates.....	470
Type of interpolation between the starting and end position.....	472
Resetting the TCPM FUNCTION.....	473

12.6 Three-dimensional tool compensation (Option #9).....474

Introduction.....	474
Definition of a normalized vector.....	475
Permissible tool shapes.....	476
Using other tools: Delta values.....	476
3-D compensation without TCPM.....	476
Face Milling: 3D compensation with TCPM.....	477
Peripheral Milling: 3-D radius compensation with TCPM and radius compensation (RL/RR).....	478

13 Programming: Pallet editor..... 481

13.1 Pallet management..... 482

 Application..... 482

 Select pallet table..... 484

 Exiting the pallet file..... 484

 Run pallet file..... 484

14 Programming: Turning Operations.....	487
14.1 Turning Operations on Milling Machines (Option #50).....	488
Introduction.....	488
14.2 Basic functions (Option 50).....	489
Switching between milling/turning mode of operation.....	489
Graphical display of turning operations.....	491
Program spindle speed.....	492
Feed rate.....	493
14.3 Unbalance functions (Option #50).....	494
Unbalance while turning.....	494
Measure Unbalance cycle.....	496
14.4 Tools in turning mode (Option #50).....	497
Tool call.....	497
Tool compensation in the program.....	498
Tool data.....	499
Tool tip radius compensation TRC.....	504
14.5 Turning program functions (Option #50).....	505
Recessing and undercutting.....	505
Blank form update TURNDATA BLANK.....	511
Inclined turning.....	512

15 Manual operation and setup.....	515
15.1 Switch-on, switch-off.....	516
Switch-on.....	516
Switch-off.....	518
15.2 Moving the machine axes.....	519
Note.....	519
Moving the axis with the machine axis direction buttons.....	519
Incremental jog positioning.....	519
Traverse with electronic handwheels.....	520
15.3 Spindle speed S, feed rate F and miscellaneous function M.....	530
Application.....	530
Entering values.....	530
Adjusting spindle speed and feed rate.....	531
Activating feed-rate limitation.....	531
15.4 Optional safety concept (Functional safety FS).....	532
Miscellaneous.....	532
Explanation of terms.....	533
Checking the axis positions.....	534
Activating feed-rate limitation.....	534
Additional status displays.....	535
15.5 Datum management with the preset table.....	536
Note.....	536
Saving the datums in the preset table.....	537
Activating the datum.....	542
15.6 Datum setting without a 3-D touch probe.....	543
Note.....	543
Preparation.....	543
Setting datum with an end mill.....	543
Using touch probe functions with mechanical probes or measuring dials.....	544

15.7 Using 3-D touch probes..... 545

Overview.....	545
Functions in touch probe cycles.....	547
Selecting touch probe cycles.....	549
Recording measured values from the touch-probe cycles.....	550
Writing measured values from the touch probe cycles in a datum table.....	551
Writing measured values from the touch probe cycles in the preset table.....	552

15.8 Calibrating a 3-D touch trigger probe..... 553

Introduction.....	553
Calibrating the effective length.....	554
Calibrating the effective radius and compensating center misalignment.....	555
Displaying calibration values.....	558

15.9 Compensating workpiece misalignment with 3-D touch probe..... 559

Introduction.....	559
Identifying basic rotation.....	560
Saving a basic rotation in the preset table.....	560
Compensation of workpiece misalignment by rotating the table.....	560
Displaying a basic rotation.....	561
Canceling a basic rotation.....	561
Measuring 3-D basic rotation.....	561

15.10 Datum setting with 3-D touch probe..... 563

Overview.....	563
Datum setting in any axis.....	563
Corner as datum.....	564
Circle center as datum.....	565
Setting a center line as datum.....	567
Measuring workpieces with a 3-D touch probe.....	568

15.11 Tilting the working plane (Option #8)..... 571

Application, function.....	571
Traversing reference points in tilted axes.....	573
Position display in a tilted system.....	573
Limitations on working with the tilting function.....	573
To activate manual tilting:.....	574
Setting the current tool-axis direction as the active machining direction.....	575
Setting the datum in a tilted coordinate system.....	576

16 Positioning with Manual Data Input.....577

16.1 Programming and executing simple machining operations.....578

 Positioning with manual data input (MDI)..... 578

 Protecting and erasing programs in \$MDI.....581

17 Test run and program run..... 583

17.1 Graphics.....584

Application.....	584
Speed of the Setting test runs.....	585
Overview: Display modes.....	586
3-D view.....	587
Plan view.....	590
Projection in three planes.....	590
Repeating graphic simulation.....	592
Tool display.....	592
Measurement of machining time.....	593

17.2 Showing the workpiece blank in the working space.....594

Application.....	594
------------------	-----

17.3 Functions for program display..... 595

Overview.....	595
---------------	-----

17.4 Test Run.....596

Application.....	596
------------------	-----

17.5 Program run.....598

Application.....	598
Running a part program.....	599
Interrupt machining.....	600
Moving the machine axes during an interruption.....	601
Resuming program run after an interruption.....	601
Retraction after a power interruption.....	603
Any entry into program (mid-program startup).....	605
Returning to the contour.....	607

17.6 Automatic program start.....608

Application.....	608
------------------	-----

17.7 Optional block skip..... 609

Application.....	609
Inserting the "/" character.....	609
Erasing the "/" character.....	609

17.8 Optional program-run interruption.....610

 Application..... 610

18 MOD functions.....	611
18.1 MOD function.....	612
Selecting MOD functions.....	612
Changing the settings.....	612
Exiting MOD functions.....	612
Overview of MOD functions.....	613
18.2 Graphic settings.....	614
18.3 Machine settings.....	615
External access.....	615
Entering traverse limits.....	617
Tool usage file.....	618
Select kinematics.....	619
18.4 System settings.....	620
Set the system time.....	620
18.5 Select the position display.....	621
Application.....	621
18.6 Setting the unit of measure.....	622
Application.....	622
18.7 Displaying operating times.....	622
Application.....	622
18.8 Software numbers.....	623
Application.....	623
18.9 Entering the code number.....	623
Application.....	623

18.10 Setting up data interfaces.....	624
Serial interfaces on the TNC 640.....	624
Application.....	624
Setting the RS-232 interface.....	624
Setting the BAUD RATE (baudRate).....	624
Setting the protocol (protocol).....	625
Setting data bits (dataBits).....	625
Check parity (parity).....	625
Setting the stop bits (stopBits).....	625
Setting handshaking (flowControl).....	626
File system for file operations (fileSystem).....	626
Block Check Character (bccAvoidCtrlChar).....	626
Condition of RTS line (rtsLow).....	626
Define behavior after reception of ETX (noEotAfterEtx).....	626
Settings for data transfer with the TNCserver PC software.....	627
Setting the operating mode of the external device (fileSystem).....	628
Data transfer software.....	629
18.11 Ethernet interface.....	631
Introduction.....	631
Connection options.....	631
Configuring the TNC.....	631
18.12 Firewall.....	637
Application.....	637
18.13 Configure HR 550 FS wireless handwheel.....	640
Application.....	640
Assigning the handwheel to a specific handwheel holder.....	640
Setting the transmission channel.....	641
Selecting the transmitter power.....	641
Statistical data.....	642
18.14 Load machine configuration.....	643
Application.....	643

19 Tables and overviews.....	645
19.1 Machine-specific user parameters.....	646
Application.....	646
19.2 Connector pin layout and connection cables for data interfaces.....	658
RS-232-C/V.24 interface for HEIDENHAIN devices.....	658
Non-HEIDENHAIN devices.....	660
Ethernet interface RJ45 socket.....	661
19.3 Technical Information.....	662
19.4 Overview tables.....	670
Fixed cycles.....	670
Miscellaneous functions.....	671
19.5 Functions of the TNC 640 and the iTNC 530 compared.....	673
Comparison: Specifications.....	673
Comparison: Data interfaces.....	673
Comparison: Accessories.....	674
Comparison: PC software.....	674
Comparison: Machine-specific functions.....	675
Comparison: User Functions.....	675
Comparator: Cycles.....	682
Comparison: Miscellaneous functions.....	685
Comparison: Touch probe cycles in the Manual Operation and El. Handwheel modes.....	687
Comparison: Touch probe cycles for automatic workpiece inspection.....	688
Comparison: Differences in programming.....	689
Comparison: Differences in Test Run, functionality.....	694
Comparison: Differences in Test Run, operation.....	694
Comparison: Differences in Manual Operation, functionality.....	694
Comparison: Differences in Manual Operation, operation.....	696
Comparison: Differences in Program Run, operation.....	696
Comparison: Differences in Program Run, traverse movements.....	697
Comparison: Differences in MDI operation.....	701
Comparison: Differences in programming station.....	702

1

**First Steps with
the TNC 640**

1 First Steps with the TNC 640

1.1 Overview

1.1 Overview

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

The following topics are included in this chapter:

- Machine switch-on
- Programming the first part
- Graphically testing the first part
- Setting up tools
- Workpiece setup
- Running the first program

1.2 Machine switch-on

Acknowledging the power interruption and moving to the reference points



Switch-on and crossing over the reference points can vary depending on the machine tool. Refer to your machine manual.

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



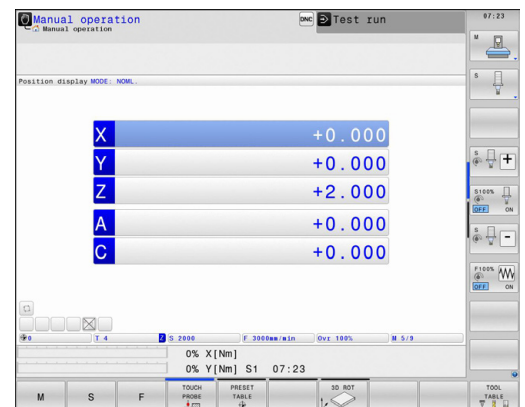
- ▶ Press the CE key: The TNC compiles the PLC program



- ▶ Switch on the control voltage: The TNC checks operation of the emergency stop circuit and goes into the reference run mode



- ▶ Cross the reference points manually in the displayed sequence: For each axis press the machine **START** button. If you have absolute linear and angle encoders on your machine there is no need for a reference run



The TNC is now ready for operation in the **Manual Operation** mode.

Further information on this topic

- Traversing the reference marks: see "Switch-on", page 516
- Operating modes: see "Programming", page 76

1.3 Programming the first part

Selecting the correct operating mode

You can write programs only in Programming mode:








- Press the Programming operating mode key: The TNC switches to **Programming mode**

Further information on this topic

- Operating modes: see "Programming", page 76

The most important TNC keys

Key	Functions for conversational guidance
	Confirm entry and activate the next dialog prompt
	Ignore the dialog question
	End the dialog immediately
	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 programs: see "Editing a program", page 109
- Overview of keys: see "Controls of the TNC", page 2

First Steps with the TNC 640

1.3 Programming the first part

Opening a new program/file management

PGM
MGT

- Press the PGM MGT key: The TNC opens the file manager. The file management of the TNC is arranged much like the file management on a PC with the Windows Explorer. The file management enables you to manage data on the internal memory of the TNC

- Use the arrow keys to select the folder in which you want to open the new file

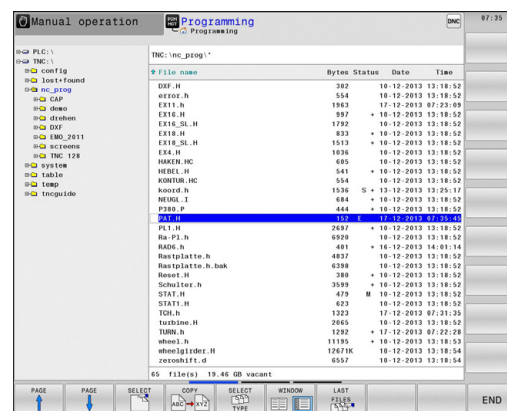
- Enter any desired file name with the extension **.H**

ENT

- Confirm with the **ENT** key: The control asks you for the unit of measurement for the new program

MM

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



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

Further information on this topic

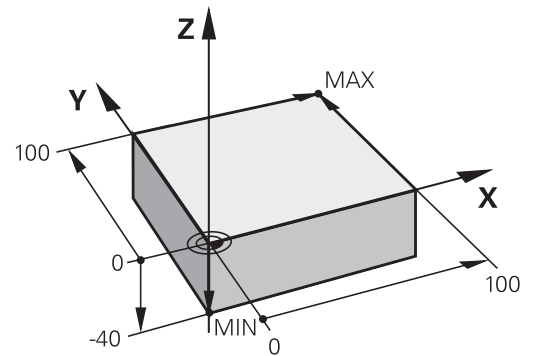
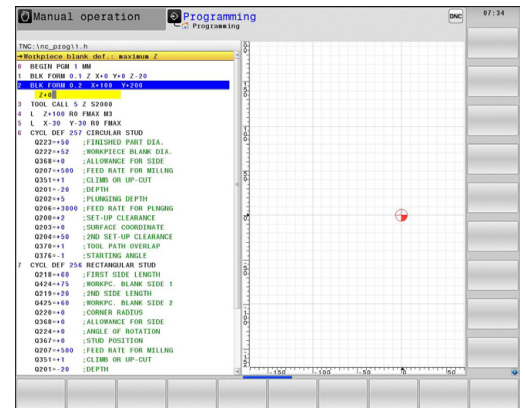
- File Management: see "Working with the File Manager", page 117
- Creating a new program: see "Opening programs and entering", page 101

Defining a workpiece blank

After you have created a new 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 reference point.

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

- ▶ **Working plane in graphic: XY?:** Enter the active spindle axis. Z 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 reference point, e.g. 0, confirm with the **ENT** key
- ▶ **Workpiece blank def.: Minimum Y:** Smallest Y coordinate of the workpiece blank with respect to the reference point, e.g. 0. Confirm with the **ENT** key
- ▶ **Workpiece blank def.: Minimum Z:** Smallest Z coordinate of the workpiece blank with respect to the reference point, 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 reference point, 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 reference point, 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 reference point, e.g. 0. Confirm with the **ENT** key. The TNC concludes the dialog



Example NC blocks

```
0 BEGIN PGM NEW MM
```

```
1 BLK FORM 0.1 Z X+0 Y+0 Z-40
```

```
2 BLK FORM 0.2 X+100 Y+100 Z+0
```

```
3 END PGM NEW MM
```

Further information on this topic

- Defining the workpiece blank: page 104

First Steps with the TNC 640

1.3 Programming the first part

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

- 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 program

Further information on this topic

- Contour programming: see "Tool movements in the program"

Recommended program layout for simple cycle programs

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

Further information on this topic

- Cycle programming: See User's Manual for Cycles

Layout of contour machining programs

```

0 BEGIN PGM BSPCONT MM
1 BLK FORM 0.1 Z X... Y... Z...
2 BLK FORM 0.2 X... Y... Z...
3 TOOL CALL 5 Z S5000
4 L Z+250 R0 FMAX
5 L X... Y... R0 FMAX
6 L Z+10 R0 F3000 M13
7 APPR ... RL F500
...
16 DEP ... X... Y... F3000 M9
17 L Z+250 R0 FMAX M2
18 END PGM BSPCONT MM

```

Cycle program layout

```

0 BEGIN PGM BSBCYC MM
1 BLK FORM 0.1 Z X... Y... Z...
2 BLK FORM 0.2 X... Y... Z...
3 TOOL CALL 5 Z S5000
4 L Z+250 R0 FMAX
5 PATTERN DEF POS1( X... Y...
  Z... ) ...
6 CYCL DEF...
7 CYCL CALL PAT FMAX M13
8 L Z+250 R0 FMAX M2
9 END PGM BSBCYC MM

```


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 TNC in the screen header.



- ▶ Call the tool: Enter the tool data. Confirm each of your entries with the **ENT** key. Do not forget the tool axis **Z**



- ▶ 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
- ▶ Confirm **Radius comp.: RL/RR/no comp?** by pressing the **ENT** key: Do not activate the radius compensation
- ▶ Confirm **Feed rate F=?** with the **ENT** key: Move at rapid traverse (**FMAX**)
- ▶ Enter **Miscellaneous function M?** and confirm with the **END** key: The TNC saves the input positioning block



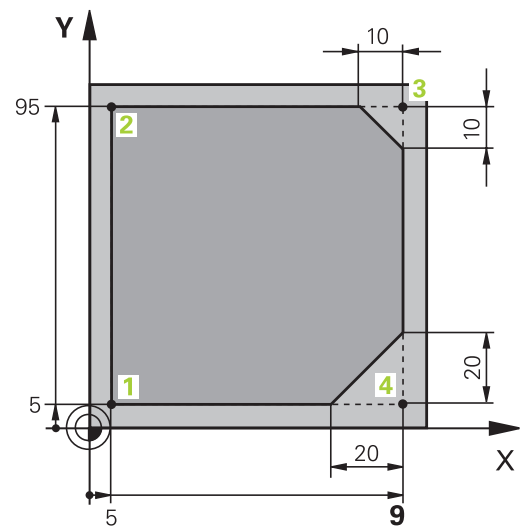
- ▶ Preposition the tool in the working plane: Press the orange **X** axis key and enter the value for the position to be approached, e.g. -20
- ▶ Press the orange axis key **Y** and enter the value for the position to be approached, e.g. -20. Confirm your entry with the **ENT** key.
- ▶ Confirm **Radius comp.: RL/RR/no comp?** by pressing the **ENT** key: Do not activate the radius compensation
- ▶ Confirm **Feed rate F=?** with the **ENT** key: Move at rapid traverse (**FMAX**)



- ▶ Confirm **Miscellaneous function M?** with the **END** key: The TNC saves the entered positioning block
- ▶ 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
- ▶ Confirm **Radius comp.: RL/RR/no comp?** by pressing the **ENT** key: Do not activate the radius compensation

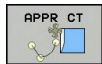


- ▶ **Feed rate F=?** Enter the positioning feed rate, e.g. 3000 mm/min, confirm with the **ENT** key
- ▶ **Miscellaneous function M?** Switch on the spindle and coolant, e.g. **M13**, confirm with the **END** key: The TNC saves the entered positioning block
- ▶ Move to the contour: Press the **APPR/DEP** key: The TNC shows a soft-key row with approach and departure functions



First Steps with the TNC 640

1.3 Programming the first part



- ▶ Select the approach function **APPR CT**: Enter the coordinates of the contour starting point **1** in X and Y, e.g. 5/5, confirm with the **ENT** key
- ▶ **Center angle?** Enter the approach angle, e.g. 90°, confirm with the **ENT** key
- ▶ **Circle radius?** Enter the circular radius, e.g. 8 mm, confirm with the **ENT** key
- ▶ Confirm the **Radius comp.: RL/RR/no comp?** with the **RL** soft key: Activate the radius compensation to the left of the programmed contour
- ▶ **Feed rate F=?** Enter the machining feed rate, e.g. 700 mm/min, save your entry 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 the chamfer at contour point **3**: Enter the chamfer width 10 mm and save with the **END** key



- ▶ Move to contour point **4**: Enter the Y coordinate 5 and save your entry with the **END** key



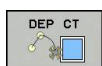
- ▶ Define the chamfer at contour point **4**: Enter the chamfer width 20 mm and save with the **END** key



- ▶ Move to contour point **1**: Enter the X coordinate 5 and save your entry with the **END** key



- ▶ Depart the contour



- ▶ Select the departure function **DEP CT**
- ▶ **Center angle?** Enter the departure angle, e.g. 90°, confirm with the **ENT** key
- ▶ **Circle radius?** Enter the departure radius, e.g. 8 mm, confirm with the **ENT** key
- ▶ **Feed rate F=?** Enter the positioning feed rate, e.g. 3000 mm/min, confirm with the **ENT** key
- ▶ **Miscellaneous function M?** Switch on the coolant, e.g. **M9**, confirm with the **END** key: The TNC saves the entered positioning block



- ▶ Retracting tool: Press the orange axis key **Z** and enter the value for the position to be approached, e.g. 250. Press the **ENT** key
- ▶ Confirm **Radius comp.: RL/RR/no comp?** by pressing the **ENT** key: Do not activate the radius compensation
- ▶ Confirm **Feed rate F=?** with the **ENT** key: Move at rapid traverse (**FMAX**)
- ▶ **MISCELLANEOUS FUNCTION M? ENTER M2** to end the program and confirm with the **END** key: The TNC saves the entered positioning block

Further information on this topic

- **Complete example with NC blocks:** see "Example: Linear movements and chamfers with Cartesian coordinates", page 231
- Creating a new program: see "Opening programs and entering", page 101
- Approaching/departing contours: see " Approaching and departing a contour", page 212
- Programming contours: see "Overview of path functions", page 222
- Programmable feed rates: see "Possible feed rate input", page 107
- Tool radius compensation: see "Tool radius compensation ", page 201
- Miscellaneous functions (M): see "M functions for program run inspection, spindle and coolant ", page 369

First Steps with the TNC 640

1.3 Programming the first part

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.



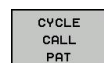
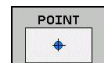
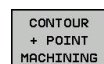
- ▶ Call the tool: Enter the tool data. Confirm each of your entries with the **ENT** key. Do not forget the tool axis



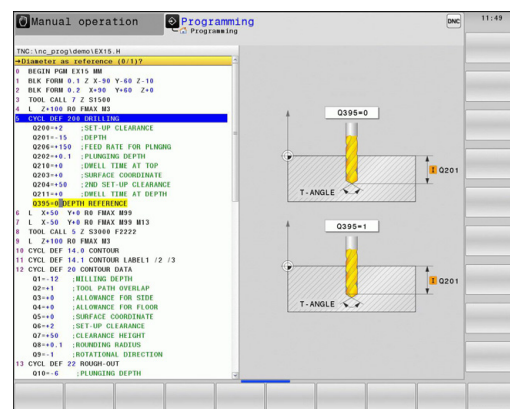
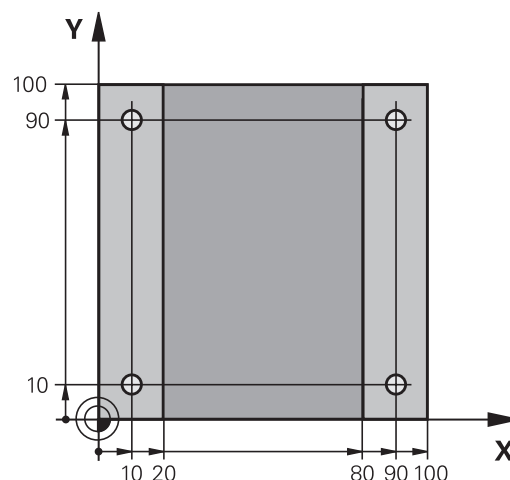
- ▶ Retract tool: Press the orange axis key **Z** and enter the value for the position to be approached, e.g. 250. Press the **ENT** key
- ▶ Confirm **Radius comp.:** **RL/RR/no comp.?** by pressing the **ENT** key: Do not activate radius compensation
- ▶ Confirm **Feed rate F=?** with the **ENT** key: Move at rapid traverse (**FMAX**)
- ▶ **Miscellaneous function M?** Confirm with the **END** key: The TNC saves the entered positioning block
- ▶ Call the cycle menu



- ▶ Display the drilling cycles
- ▶ Select the standard drilling cycle 200: The TNC starts the dialog for cycle definition. Enter all parameters requested by the TNC step by step and conclude each entry with the **ENT** key. In the screen to the right, the TNC also displays a graphic showing the respective cycle parameter
- ▶ Call the menu for special functions



- ▶ Display the functions for point machining
- ▶ Select the pattern definition
- ▶ Select point entry: Enter the coordinates of the 4 points and confirm each with the **ENT** key. After entering the fourth point, save the block with the **END** key
- ▶ Display the menu for defining the cycle call
- ▶ Run the drilling cycle on the defined pattern:
- ▶ Confirm **Feed rate F=?** with the **ENT** key: Move at rapid traverse (**FMAX**)
- ▶ **Miscellaneous function M?** Switch on the spindle and coolant, e.g. **M13**. Confirm with the **END** key: The TNC saves the entered positioning block





- ▶ Enter 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
- ▶ Confirm **Radius comp.: RL/RR/no comp.?** by pressing the **ENT** key: Do not activate radius compensation
- ▶ Confirm **Feed rate F=?** with the **ENT** key: Move at rapid traverse (**FMAX**)
- ▶ **Miscellaneous function M?** Enter **M2** to end the program and confirm with the **END** key: The TNC saves the entered positioning block

Example NC blocks

0 BEGIN PGM C200 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-40	Definition of workpiece blank
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 5 Z S4500	Tool call
4 L Z+250 R0 FMAX	Retract the tool
5 PATTERN DEF POS1 (X+10 Y+10 Z+0) POS2 (X+10 Y+90 Z+0) POS3 (X+90 Y+90 Z+0) POS4 (X+90 Y+10 Z+0)	Define the machining positions
6 CYCL DEF 200	Define the cycle
Q200=2 ;SET-UP CLEARANCE	
Q201=-20 ;DEPTH	
Q206=250 ;FEED RATE FOR PLNGNG	
Q202=5 ;	
Q210=0 ;DWELL TIME AT TOP	
Q203=-10 ;SURFACE COORDINATE	
Q204=20 ;2ND SET-UP CLEARANCE	
Q211=0.2 ;DWELL TIME AT DEPTH	
7 CYCL CALL PAT FMAX M13	Spindle and coolant on, call the cycle
8 L Z+250 R0 FMAX M2	Retract the tool, end program
9 END PGM C200 MM	

Further information on this topic

- Creating a new program: see "Opening programs and entering", page 101
- Cycle programming: See User's Manual for Cycles, "Cycle fundamentals / Overviews"

First Steps with the TNC 640

1.4 Graphically testing the first part

1.4 Graphically testing the first part

Selecting the correct operating mode

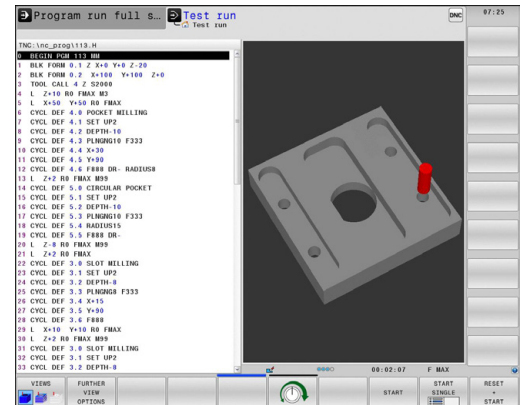
You can test programs in the **Test Run** mode:



- ▶ Press the **Test Run** operating mode key: the TNC switches to that mode

Further information on this topic

- Operating modes of the TNC: see "Modes of Operation", page 75
- Testing programs: see "Test Run", page 596



Selecting the tool table for the test run

You only need to execute this step if you have not activated a tool table in the **Test Run** mode.



- ▶ Press the **PGM MGT** key: The TNC opens the file manager



- ▶ Press the **SELECT TYPE** soft key: The TNC shows a soft-key menu for selection of the file type to be displayed



- ▶ Press the **DEFAULT** soft key: The TNC shows all saved files in the right window



- ▶ Move the highlight to the left onto the directories



- ▶ Move the highlight to the **TNC:\table** directory



- ▶ Move the highlight to the right onto the files



- ▶ Move the highlight to the file **TOOL.T** (active tool table) and load with the **ENT** key: **TOOL.T** receives the status **S** and is therefore active for the test run



- ▶ Press the **END** key: Exit the file manager

Further information on this topic

- Tool management: see "Enter tool data into the table", page 174
- Testing programs: see "Test Run", page 596

Choosing the program you want to test



- ▶ Press the **PGM MGT** key: The TNC opens the file manager



- ▶ Press the **LAST FILES** soft key: The TNC opens a pop-up window with the most recently selected files
- ▶ Use the arrow keys to select the program that you want to test. Load with the ENT key

Further information on this topic

- Selecting a program: see "Working with the File Manager", page 117

Selecting the screen layout and the view



- ▶ Press the key for selecting the screen layout. The TNC shows all available alternatives in the soft-key row



- ▶ Press the **PROGRAM + GRAPHICS** soft key: In the left half of the screen the TNC shows the program; in the right half it shows the workpiece blank



- ▶ Press the **FURTHER VIEW OPTIONS** soft key



- ▶ Shift the soft-key row and select the desired view by soft key

The TNC features the following views:

Soft keys	Function
	Volume view
	Volume view and tool paths
	Tool paths

Further information on this topic

- Graphic functions: see "Graphics ", page 584
- Running a test run: see "Test Run", page 596

First Steps with the TNC 640

1.4 Graphically testing the first part

Starting the test run



- ▶ Press the **RESET + START** soft key: The TNC simulates the active program up to a programmed break or to the program end
- ▶ While the simulation is running, you can use the soft keys to change views



- ▶ Press the **STOP** soft key: The TNC interrupts the test run



- ▶ Press the **START** soft key: The TNC resumes the test run after a break

Further information on this topic

- Running a test run: see "Test Run", page 596
- Graphic functions: see "Graphics ", page 584
- Adjust the simulation speed: see "Speed of the Setting test runs", page 585

1.5 Setting up tools

Selecting the correct operating mode

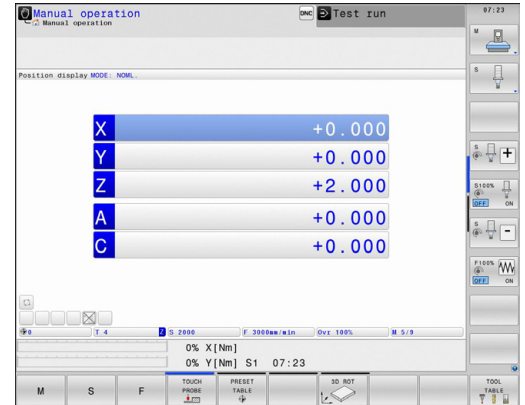
Tools are set up in the **Manual Operation** mode:



- Press the operating-mode key: The TNC switches to the **Manual** mode of operation

Further information on this topic

- Operating modes of the TNC: see "Modes of Operation", page 75



Preparing and measuring tools

- Clamp the required tools in their tool holders
- When measuring with an external tool presetter: Measure the tools, note down the length and radius, or transfer them directly to the machine through a transfer program
- When measuring on the machine: store the tools in the tool changer, see page 67

The pocket table TOOL_PTCH



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

In the pocket table TOOL_PTCH (permanently saved under **TNC: \TABLE**) you specify which tools your tool magazine contains.

To enter data in the pocket table TOOL_PTCH, proceed as follows:



- Display the tool table: The TNC shows the tool table



- Display the pocket table: The TNC shows the pocket table
- Edit the pocket table: Set the **EDIT** soft key to ON
- With the upward or downward arrow keys you can select the pocket number that you want to edit
- With the rightward or leftward arrow keys you can select the data that you want to edit
- Exit the pocket table: press the **END** key.

P	T	TNAME	RSV	ST	F	L	DOC
0.0	5.010						
1.1	1.02						
1.2	2.04						
1.3	3.06						
1.4	4.08						
1.5	5.010						
1.6	6.012						
1.7	7.014						
1.8	8.016						
1.9	9.018						
1.10	10.020						
1.11	11.022						
1.12	12.024						
1.13	13.026						
1.14	14.028						
1.15	15.030						
1.16	16.032						
1.17	17.034						
1.18	18.036						
1.19	19.038						
1.20	20.040						
1.21	21.042						
1.22	22.044						
1.23	23.046						
1.24	24.048						
1.25	25.050						
1.26	26.052						

Further information on this topic

- Operating modes of the TNC: see "Modes of Operation", page 75
- Working with the pocket table: see "Pocket table for tool changer", page 183

First Steps with the TNC 640

1.6 Workpiece setup

1.6 Workpiece setup

Selecting the correct operating mode

Workpieces are set up in the **or** mode



- ▶ Press the operating-mode key: The TNC switches to the **Manual** mode of operation

Further information on this topic

- Operating mode : see "Moving the machine axes", page 519

Clamping the workpiece

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

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

Further information on this topic

- Setting datums with 3-D touch probe: see "Datum setting with 3-D touch probe", page 563
- Setting datums without 3-D touch probe: see "Datum setting without a 3-D touch probe", page 543

Datum setting with 3-D touch probe

- Insert a 3-D touch probe: In the **Positioning with Manual Data Input** mode, run a **TOOL CALL** block containing the tool axis and then return to the **Manual Operation** mode



- Select the probing functions: The TNC displays the available functions in the soft-key row



- Set the datum at a workpiece corner, for example
- Position the touch probe near the first touch point on the first workpiece edge
- Select the probing direction via soft key
- Press NC start: The touch probe moves in the defined direction until it contacts the workpiece and then automatically returns to its starting point
- Use the axis-direction keys to pre-position the touch probe to a position near the second touch point on the first workpiece edge
- Press NC start: The touch probe moves in the defined direction until it contacts the workpiece and then automatically returns to its starting point
- Use the axis-direction keys to pre-position the touch probe to a position near the first touch point on the second workpiece edge
- Select the probing direction via soft key
- Press NC start: The touch probe moves in the defined direction until it contacts the workpiece and then automatically returns to its starting point
- Use the axis-direction keys to pre-position the touch probe to a position near the second touch point on the second workpiece edge
- Press NC start: The touch probe moves in the defined direction until it contacts the workpiece and then automatically returns to its starting point
- Then the TNC shows the coordinates of the measured corner point



- To set to 0: Press the **SET DATUM** soft key
- Press the **END** soft key to close the menu

Further information on this topic

- Datum setting: see "Datum setting with 3-D touch probe ", page 563

First Steps with the TNC 640

1.7 Running the first program

1.7 Running the first program

Selecting the correct operating mode

You can run programs either in the **Single Block** or the **Full Sequence** mode:



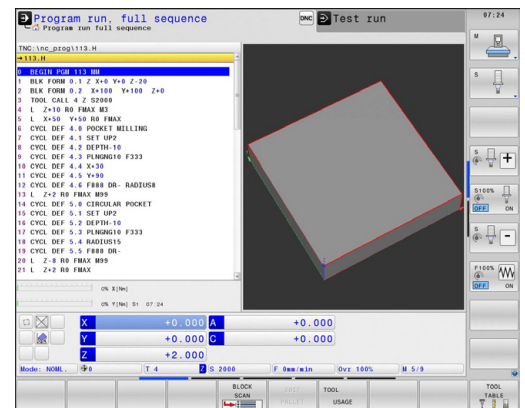
- ▶ Press the operating mode key: The TNC goes into the **Program Run, Single Block** mode and the TNC executes the program block by block. You have to confirm each block with the NC start key



- ▶ Press the **Program Run, Full Sequence** operating mode key: The TNC switches to that mode and runs the program after NC start up to a program interruption or to the end of the program

Further information on this topic

- Operating modes of the TNC: see "Modes of Operation", page 75
- Running programs: see "Program run", page 598



Choosing the program you want to run



- ▶ Press the **PGM MGT** key: The TNC opens the file manager



- ▶ Press the **LAST FILES** soft key: The TNC opens a pop-up window with the most recently selected files
- ▶ If desired, use the arrow keys to select the program that you want to run. Load with the ENT key

Further information on this topic

- File Management: see "Working with the File Manager", page 117

Start the program



- ▶ Press the NC start key: The TNC runs the active program

Further information on this topic

- Running programs: see "Program run", page 598

2

Introduction

Introduction

2.1 The TNC 640

2.1 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 an easy-to-use conversational programming language. They are designed for milling, drilling and boring machines, as well as machining centers, with up to 18 axes. You can also change the angular position of the spindle under program control.

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

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



Programming: In HEIDENHAIN conversational and DIN/ISO

The HEIDENHAIN conversational programming format is an especially easy method of writing programs. Interactive graphics illustrate the individual machining steps for programming the contour. If a production drawing is not dimensioned for NC, the FK free contour programming feature performs the necessary calculations automatically. Workpiece machining can be graphically simulated either during or before actual machining.

It is also possible to program in ISO format or DNC mode.

You can also enter and test one program while the control is running another.

Compatibility

Machining programs created on HEIDENHAIN contouring controls (starting from the TNC 150 B) may not always run on the TNC 640. If NC blocks contain invalid elements, the TNC will mark them 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, see "Functions of the TNC 640 and the iTNC 530 compared", page 673.

2.2 Visual display unit and operating panel

Display screen

The TNC is shipped with a 19-inch TFT flat-panel display.

1 Header

When the TNC is on, the selected operating modes are shown in the screen header: the machining mode at the left and the programming mode at right. The currently active operating mode is displayed in the larger box, where the dialog prompts and TNC messages also appear (unless the TNC is showing only graphics).

2 Soft keys

In the footer the TNC 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 highlighted.

3 Soft-key selection keys

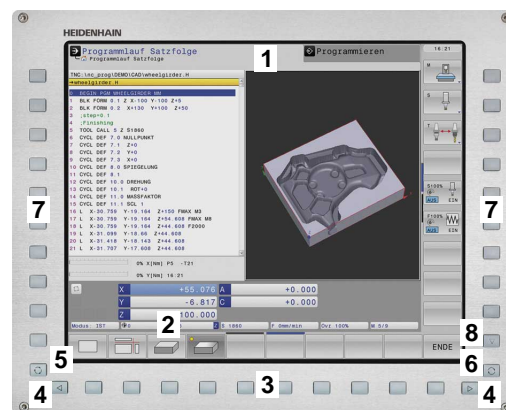
4 Keys for switching the soft keys

5 Setting the screen layout

6 Shift key for switchover between machining and programming modes

7 Soft-key selection keys for machine tool builders

8 Keys for switching the soft keys for machine tool builders



Setting the screen layout

You select the screen layout yourself: In the **Programming** mode of operation, for example, you can have the TNC show 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 program blocks in one large window. The available screen windows depend on the selected operating mode.

To change the screen layout:



- ▶ Press the screen layout key: The soft-key row shows the available layout options, see "Modes of Operation"



- ▶ Select the desired screen layout

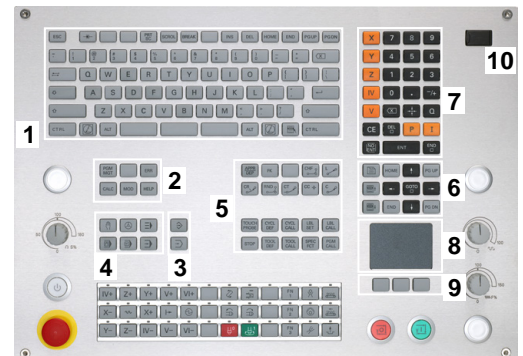
Introduction

2.2 Visual display unit and operating panel

Control Panel

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

- 1 Alpha-numeric keyboard for entering texts and file names, as well as for DIN/ISO programming
- 2
 - File management
 - Calculator
 - MOD function
 - HELP function
- 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.



Some machine manufacturers do not use the standard operating panel from HEIDENHAIN. Refer to your machine manual.

External buttons, e.g. NC START or NC STOP, are described in the manual for your machine tool.

2.3 Modes of Operation

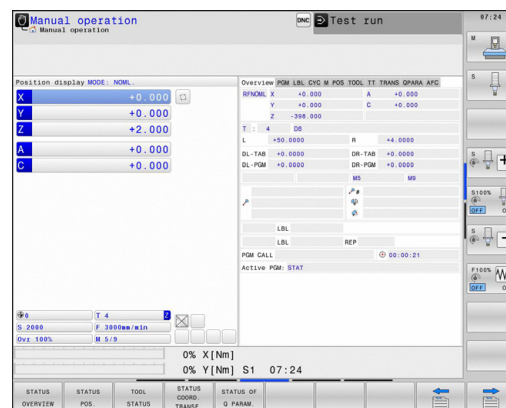
Manual Operation and El. Handwheel

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 datums and tilt the working plane.

The **El. Handwheel** mode of operation allows you to move the machine axes manually with the HR electronic handwheel.

Soft keys for selecting the screen layout (select as described previously)

Soft key	Window
POSITION	Positions
POSITION + STATUS	Left: positions, right: status display
POSITION + KINEMATICS	Left: positions, right: collision object

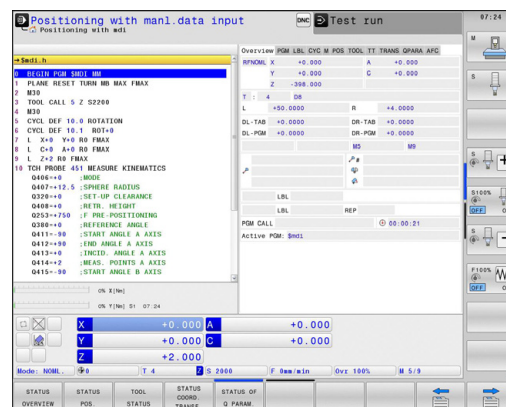


Positioning with Manual Data Input

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

Soft keys for selecting the screen layout

Soft key	Window
PGM	Program
PROGRAM + STATUS	Left: program, right: status display
POSITION + KINEMATICS	Left: program, right: collision object



Programming

In this mode of operation you can write your part 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

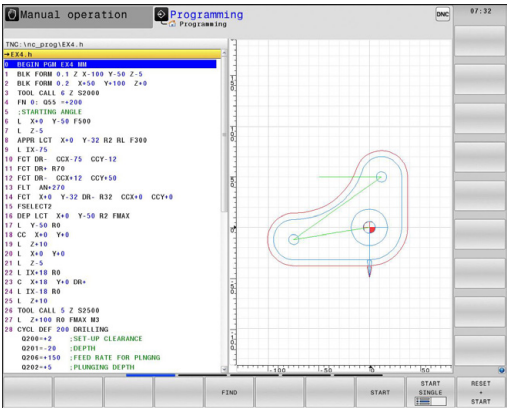
Soft key	Window
PGM	Program
PROGRAM + SECTS	Left: program, right: program structure
PROGRAM + GRAPHICS	Left: program, right: programming graphics

Test Run

In the **Test Run** mode of operation, the TNC checks programs and program sections for errors, such as geometrical incompatibilities, missing or incorrect data within the 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	Program
PROGRAM + STATUS	Left: program, right: status display
PROGRAM + GRAPHICS	Left: program, right: graphics
GRAPHICS	Graphic



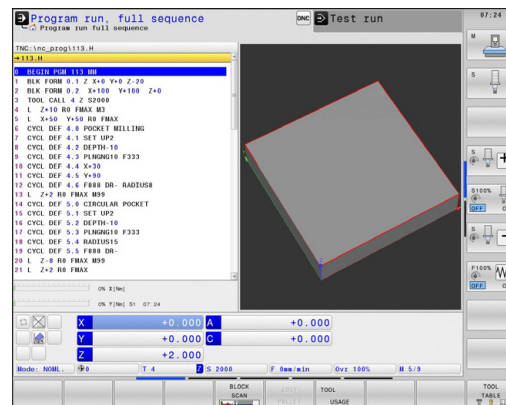
Program Run, Full Sequence and Program Run, Single Block

In the **Program run full sequence** mode of operation the TNC executes a 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** mode of operation you execute each block separately by pressing the machine START button. 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	Program
PROGRAM + STATUS	Left: program, right: status display
PROGRAM + GRAPHICS	Left: program, right: graphics
GRAPHICS	Graphic
POSITION + KINEMATICS	Left: program, right: collision object
KINEMATICS	Collision body
Soft key	Window
PALLET	Pallet table
PROGRAM + PALLET	Left: program, right: pallet table
PALLET + STATUS	Left: pallet table, right: status display
PALLET + GRAPHICS	Left: pallet table, right: graphics



2.4 Status displays







General status display

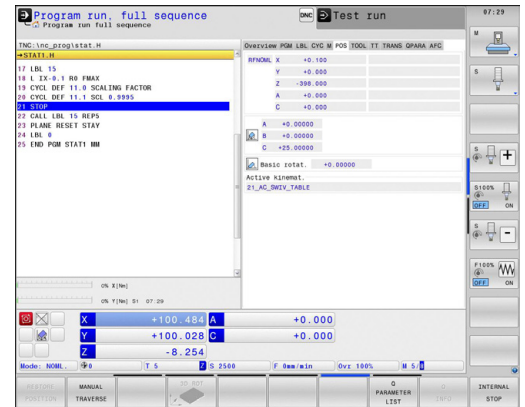
The general status display in the lower part of the screen informs you of the current state of the machine tool. It is displayed automatically in the following modes of operation:








- **Program Run, Single Block** and **Program Run, Full Sequence**, except if the screen layout is set to display only, and during
- **Positioning with Manual Data Input**.

In the **Manual Operation** and **EL. Handwheel** modes the status display appears in the large window.

Information in the status display

Icon	Meaning
ACTL.	Position display: Actual, nominal or distance-to-go coordinates mode
XYZ	Machine axes; the TNC displays auxiliary axes in lower-case letters. The sequence and quantity of displayed axes is determined by the machine tool builder. Refer to your machine manual for more information
	Number of the active presets from the preset table. If the datum was set manually, the TNC displays the text MAN behind the symbol
F S M	The displayed feed rate in inches corresponds to one tenth of the effective value. Spindle speed S, feed rate F and active M functions
	Axis is clamped
	Axis can be moved with the handwheel
	Axes are moving under a basic rotation
	Axes are moving under a 3-D basic rotation
	Axes are moving in a tilted working plane
TC PM	The M128 function or FUNCTION TCPM is active



Icon	Meaning
	No active program
	Program run has started
	Program run is stopped
	Program run is being aborted
	Turning mode is active
	The Dynamic Collision Monitoring function (DCM) is active (Option #40)
	The Adaptive Feed Function (AFC) is active (Option #45)
ACC	The Active Chatter Control feature (ACC) is active (Option #145)
CTC	The CTC function is active (Option #141)

Additional status displays

The additional status displays contain detailed information on the program run. They can be called in all operating modes except for the **Programming** mode of operation.

To switch on the additional status display



- Call the soft-key row for screen layout

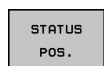


- Select the screen layout with additional status display: In the right half of the screen, the TNC shows the **OVERVIEW** status form

To select an additional status display



- Switch the soft-key rows until the STATUS soft keys appear



- Either select the additional status display directly by soft key, e.g. positions and coordinates, or



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

The available status displays described below can be selected either directly by soft key or with the switch-over soft keys.



Please note that some of the status information described below is not available unless the associated software option is enabled on your TNC.

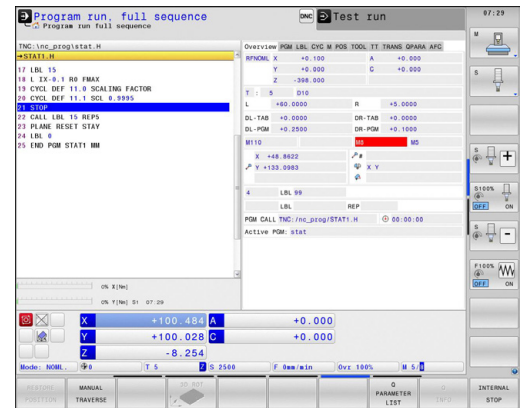
Introduction

2.4 Status displays

Overview

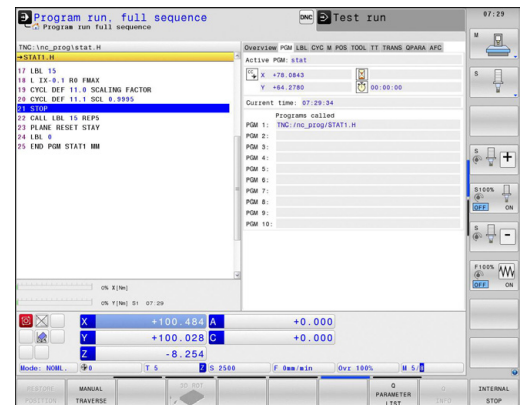
After switch-on, the TNC displays the **Overview** status form, provided that you have selected the **PROGRAM+STATUS** screen layout (or **POSITION + STATUS**). The overview form contains a summary of the most important status information, which you can also find on the various detail forms.

Soft key	Meaning
STATUS	Position display
OVERVIEW	Tool information
	Active M functions
	Active coordinate transformations
	Active subprogram
	Active program section repeat
	Program called with PGM CALL
	Current machining time
	Name of the active main program



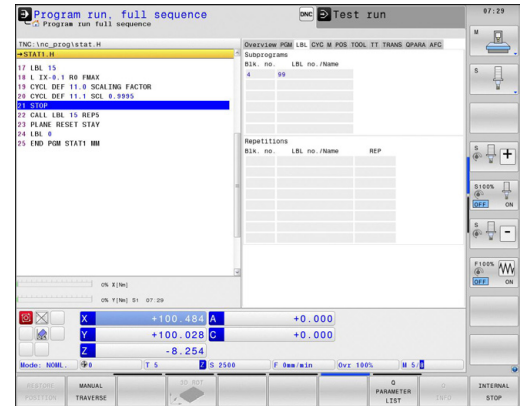
General program information (PGM tab)

Soft key	Meaning
No direct selection possible	Name of the active main program
	Circle center CC (pole)
	Dwell time counter
	Machining time when the program was completely simulated in the Test Run operating mode
	Current machining time in percent
	Current time
	Active programs



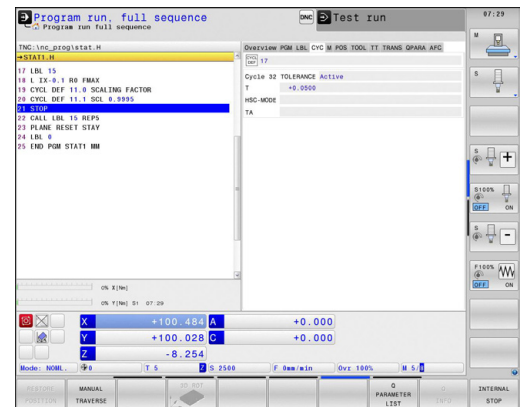
Program section repeat/Subprograms (LBL tab)

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




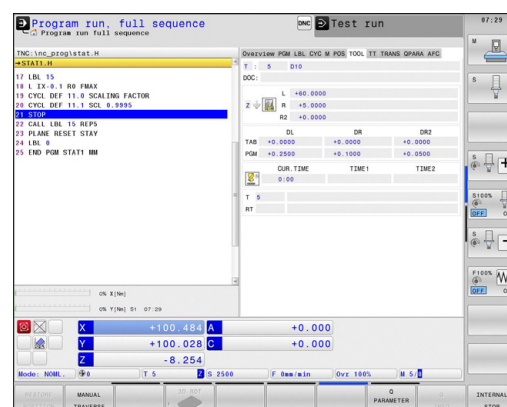
Information on standard cycles (CYC tab)

Soft key	Meaning
No direct selection possible	Active fixed cycle
	Active values of Cycle 32 Tolerance



Information on tools (TOOL tab)

Soft key	Meaning
	Display of active tool: <ul style="list-style-type: none"> ■ T: Tool number and tool name ■ RT: Number and name of a replacement tool
	Tool axis
	Tool length and tool radii
	Oversizes (delta values) from the tool table (TAB) and the TOOL CALL (PGM)
	Tool life, maximum tool life (TIME 1) and maximum tool life for TOOL CALL (TIME 2)
	Display of programmed tool and replacement tool

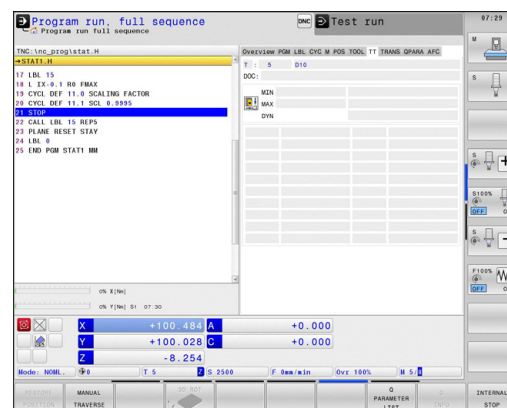


Tool measurement (TT tab)



The TNC displays the TT tab only if the function is active on your machine.

Soft key	Meaning
No direct selection possible	Number of the tool to be measured
	Display whether the tool radius or the tool length is being measured
	MIN and MAX values of the individual cutting edges and the result of measuring the rotating tool (DYN = dynamic measurement)
	Cutting edge number with the corresponding measured value. If the measured value is followed by an asterisk, the tolerance in the tool table was exceeded



Coordinate transformations (TRANS tab)

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

For further information, refer to the User's Manual for Cycles, "Coordinate Transformation Cycles."

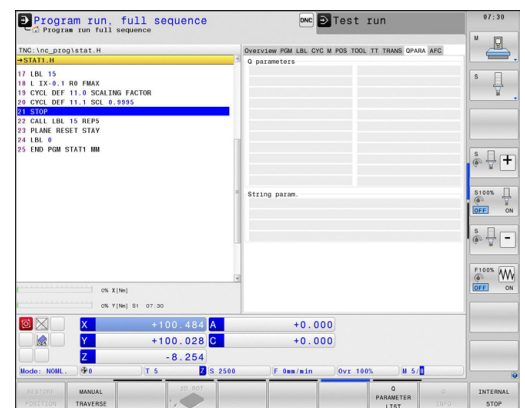
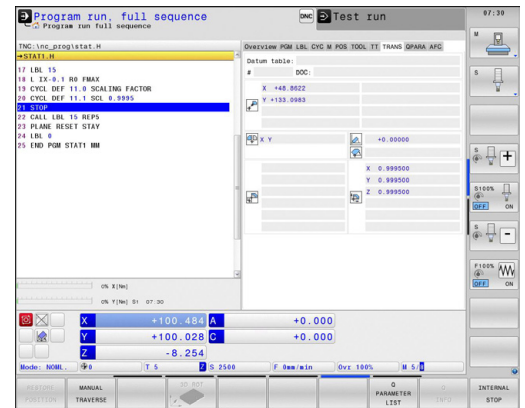
Displaying Q parameters (QPARA tab)

Soft key	Meaning
STATUS OF Q PARAM.	Display the current values of the defined Q parameters
	Display the character strings of the defined string parameters



Press the **Q PARAMETER LIST** soft key. The TNC 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} .

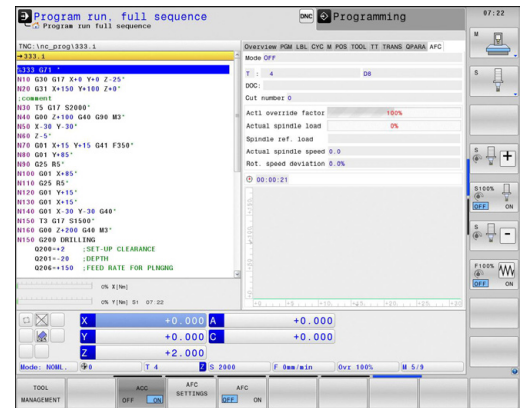


Adaptive Feed Control (AFC tab, Option #45)



The TNC displays the AFC tab only if the function is active on your machine.

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



2.5 Window Manager



The machine tool builder determines the available functions and behavior of the window manager. Refer to your machine manual.

The TNC features the Xfce window manager. Xfce is a standard application for UNIX-based operating systems, and is used to manage graphical user interfaces. The following functions are possible with the window manager:

- Display a task bar for switching between various applications (user interfaces).
- Manage an additional desktop, on which special applications from your machine tool builder can run.
- Control the focus between NC-software applications and those of the machine tool builder.
- The size and position of pop-up windows can be changed. It is also possible to close, minimize and restore the pop-up windows.



The TNC shows a star in the upper left of the screen if an application of the window manager or the window manager itself has caused an error. In this case, switch to the window manager and correct the problem. If required, refer to your machine manual.

Task bar

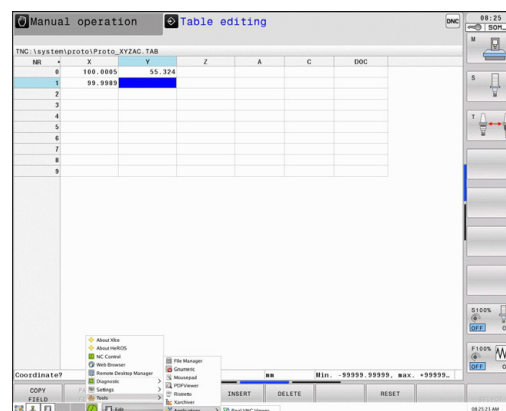
In the task bar you can choose different workspaces by mouse click. The TNC provides the following workspaces:

- Workspace 1: Active mode of operation
- Workspace 2: Active programming mode
- Workspace 3: Manufacturer's applications (optionally available)

In the task bar you can also select other applications that you have started together with the TNC (switch for example to the **PDF viewer** or **TNCguide**)

Click the green HEIDENHAIN symbol to open a menu in which you can get information, make settings or start applications. The following functions are available:

- **About HEROS**: Information about the operating system of the TNC
- **NC Control**: Start and stop the TNC software. Only permitted for diagnostic purposes
- **Web Browser**: Start Mozilla Firefox
- **Remote Desktop Manager** (Option #133): Display and remote operation of external computer units
- **Diagnostics**: Available only to authorized specialists to start diagnostic functions
- **Settings**: Configuration of miscellaneous settings
 - **Date/Time**: Set the date and time
 - **Language**: System dialog language setting. During startup the TNC overwrites this setting with the language setting of the machine parameter CfgLanguage
 - **Network**: Network settings of the control
 - **Screensaver**: Screensaver settings
 - **SELinux**: Security software settings for Linux-based operating systems
 - **Shares**: Settings for external network drives
 - **VNC**: Setting for external softwares that access for maintenance purposes on the control for example (**V**irtual **N**etwork **C**omputing)
 - **WindowManagerConfig**: Available only to authorized specialists for setting the window manager
 - **Firewall**: Firewall settings see "Firewall", page 637
- **Tools**: Only for authorized users. The applications available under tools can be started directly by selecting the pertaining file type in the file management of the TNC (see "File Management: Fundamentals", page 114)



Introduction

2.6 Remote Desktop Manager (Option #133)

2.6 Remote Desktop Manager (Option #133)

Introduction

The Remote Desktop Manager enables you to display external computer units on the TNC screen that are connected via Ethernet and to operate them over the TNC. You can also start programs specifically under HeROS or display web pages of an external server.

The following connection options are available:

- **Windows Terminal Server (RDP)**: Displays the desktop of a remote Windows computer on the control
- **Windows Terminal Server (RemoteFX)**: Displays the desktop of a remote Windows computer on the control
- **VNC**: Connection to an external computer (e.g. HEIDENHAIN-IPC). Displays the desktop of a remote Windows or Unix computer on the control
- **Switch-off/restart of a computer**: Available only to authorized specialists
- **World Wide Web**: Available only to authorized specialists
- **SSH**: Available only to authorized specialists
- **XDMCP**: Available only to authorized specialists
- **User-defined connection**: Available only to authorized specialists



HEIDENHAIN assures a functioning connection between HeROS 5 and the IPC 6341. HEIDENHAIN cannot guarantee the correct function of any other combinations or connections to external devices.

Configuring connections – Windows Terminal Service

Configuring an external computer



You do not need additional software for your external computer for connecting to the Windows Terminal Service.

Proceed as follows to configure the external computer, e.g. in the Windows 7 operating system:

- ▶ After pressing the Windows start button select the menu item **System control** via the task bar
- ▶ Select the **System** menu item
- ▶ Select the **Advanced system settings** menu item
- ▶ Select the **Remote** tab

- ▶ In the **Remote support** area, activate the function **Permit remote support connection with this computer**
- ▶ In the **Remote desktop** area, activate the function **Permit connections from computers on which any version of remote desktop is installed**
- ▶ Confirm the settings via the **OK** button

Configuring the TNC



Depending on the operating system of your external computer and the protocol used in accordance with this, select either **Windows Terminal Service (RDP)** or **Windows Terminal Service (RemoteFX)**.

Configure the TNC as follows:

- ▶ After pressing the green HEIDENHAIN button, select the menu item **Remote Desktop Manager** via the task bar
- ▶ Press the **New connection** button in the **Remote Desktop Manager** window
- ▶ Select the menu item **Windows Terminal Service (RDP)** or **Windows Terminal Service (RemoteFX)**
- ▶ Specify the required connection information in the **Edit connection** window

Setting	Meaning	Input
Connection name	Name of the connection in the Remote Desktop Manager	Required
Restarting after end of connection	Behavior with terminated connection: <ul style="list-style-type: none"> ■ Always restart ■ Never restart ■ Always after an error ■ Ask after an error 	Required
Automatic starting upon login	Connection automatically established during control power-up	Required
Add to favorites	Connection icon in the task bar: <ul style="list-style-type: none"> ■ Double click with left mouse button: The control starts the connection ■ Single click with left mouse button: The control changes to the desktop of the connection ■ Single click with right mouse button: The control displays the connection menu 	Required
Move to the following workspace	Number of desktop for the connection, whereby desktops 0 and 1 are reserved for the NC software	Required
Release USB mass memory	Enable access to connected USB mass memory	Required
Computer	Host name or IP address of the external computer	Required
User name	Name of the user	Required
Password	User password	Required
Windows domain	Domain of the external computer	Required
Full screen mode or user-defined window size	Size of the connection window	Required

Introduction

2.6 Remote Desktop Manager (Option #133)

Setting	Meaning	Input
Entries in the Advanced options area	Available only to authorized specialists	Optional

Configuring the connection – VNC

Configuring an external computer



You do not need an additional VNC server for your external computer for connecting to VNC.
Install and configure the VNC server, e.g. the TightVNC server, before configuring the TNC.

Configuring the TNC

Configure the TNC as follows:

- ▶ Select the **Remote Desktop Manager** menu item via the task bar
- ▶ Press the **New connection** button in the **Remote Desktop Manager** window
- ▶ Select the **VNC** menu item
- ▶ Specify the required connection information in the **Edit connection** window

Setting	Meaning	Input
Connection name	Name of the connection in the Remote Desktop Manager	Required
Restarting after end of connection	Behavior with terminated connection: <ul style="list-style-type: none"> ■ Always restart ■ Never restart ■ Always after an error ■ Ask after an error 	Required
Automatic starting upon login	Connection automatically established during control power-up	Required
Add to favorites	Connection icon in the task bar: <ul style="list-style-type: none"> ■ Double click with left mouse button: The control starts the connection ■ Single click with left mouse button: The control changes to the desktop of the connection ■ Single click with right mouse button: The control displays the connection menu 	Required
Move to the following workspace	Number of desktop for the connection, whereby desktops 0 and 1 are reserved for the NC software	Required
Release USB mass memory	Permit access to connected USB mass memory	Required
Computer	Host name or IP address of the external computer	Required
Password	Password for connecting to the VNC server	Required
Full screen mode or user-defined window size	Size of the connection window	Required
Permit further connections (share)	Enable access to the VNC server also by other VNC connections	Required

Setting	Meaning	Input
View only	The external computer cannot be operated in display mode	Required
Entries in the Advanced options area	Available only to authorized specialists	Optional

Starting and stopping the connection

Once a connection has been configured, it is shown as an icon in the Remote Desktop Manager window. Click the connection icon with the right mouse key to open a menu in which the display can be started and stopped.

Use the right DIADUR key on the keyboard to change to Desktop 3 and back to the TNC interface. You can also use the task bar to get to this desktop.

If the desktop of the external connection or the external computer is active, all inputs from the mouse and the keyboard are transmitted there.

All connections are canceled automatically when the HEROS 5 operating system is shut down. Please note, however, that only the connection is canceled, whereas the external computer or the external system is not shut down automatically.

2.7 SELinux security software

SELinux is an extension for Linux-based operating systems. SELinux is an additional security software package based on Mandatory Access Control (MAC) and protects the system against the running of unauthorized processes or functions and therefore protects against viruses and other malware.

MAC means that each action must be specifically permitted otherwise the TNC will not run it. The software is intended as protection in addition to the normal access restriction in Linux. Certain processes and actions can only be executed if the standard functions and access control of SELinux permit it.



The SELinux installation of the TNC is prepared to permit running of only those programs installed with the HEIDENHAIN NC software. Other programs cannot be run with the standard installation.

The access control of SELinux under HEROS 5 is regulated as follows:

- The TNC runs only those applications installed with the HEIDENHAIN NC software.
- Files in connection with the security of the software (SELinux system files, HEROS 5 boot files, etc.) may only be changed by programs that are selected explicitly.
- New files generated by other programs must never be executed.
- USB data carriers cannot be deselected
- There are only two processes that are permitted to execute new files:
 - Starting a software update: A software update from HEIDENHAIN can replace or change system files.
 - Starting the SELinux configuration: The configuration of SELinux is usually password-protected by your machine tool builder. Refer here to the relevant machine tool manual.



HEIDENHAIN generally recommends activating SELinux because it provides additional protection against attacks from outside.

2.8 Accessories: HEIDENHAIN 3-D Touch Probes and Electronic Handwheels

3-D touch probes

The various HEIDENHAIN 3-D touch probes enable you to:

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



All of the cycle functions (touch probe cycles and fixed cycles) are described in the Cycle Programming User's Manual. Please contact HEIDENHAIN if you require a copy of this User's Manual. ID: 892905-xx

The triggering touch probes TS 220, TS 440, TS 444, TS 640 and TS 740

These touch probes are particularly effective for automatic workpiece alignment, datum setting and workpiece measurement. The TS 220 transmits the triggering signals to the TNC via cable and is a cost-effective alternative for applications where digitizing is not frequently required.

The TS 640 (see figure) and the smaller TS 440 feature infrared transmission of the triggering signal to the TNC. This makes them highly convenient for use on machines with automatic tool changers.

Principle of operation: HEIDENHAIN triggering touch probes feature a wear resisting optical switch that generates an electrical signal as soon as the stylus is deflected. This signal is transmitted to the control, which stores the current position of the stylus as the actual value.

TT 140 tool touch probe for tool measurement

The TT 140 is a triggering 3-D touch probe for tool measurement and inspection. The TNC provides three cycles for this touch probe with which you can measure the tool length and radius either with the spindle rotating or stopped. The TT 140 features a particularly rugged design and a high degree of protection, which make it insensitive to coolants and swarf. The triggering signal is generated by a wear-resistant and highly reliable optical switch.



2.8 Accessories: HEIDENHAIN 3-D Touch Probes and Electronic Handwheels**HR electronic handwheels**

Electronic handwheels facilitate moving the axis slides precisely by hand. A wide range of traverses per handwheel revolution is available. Apart from the HR 130 and HR 150 panel-mounted handwheels, HEIDENHAIN also offers the HR 410 portable handwheel.



3

**Programming:
Fundamentals, File
Management**

3.1 Fundamentals

3.1 Fundamentals

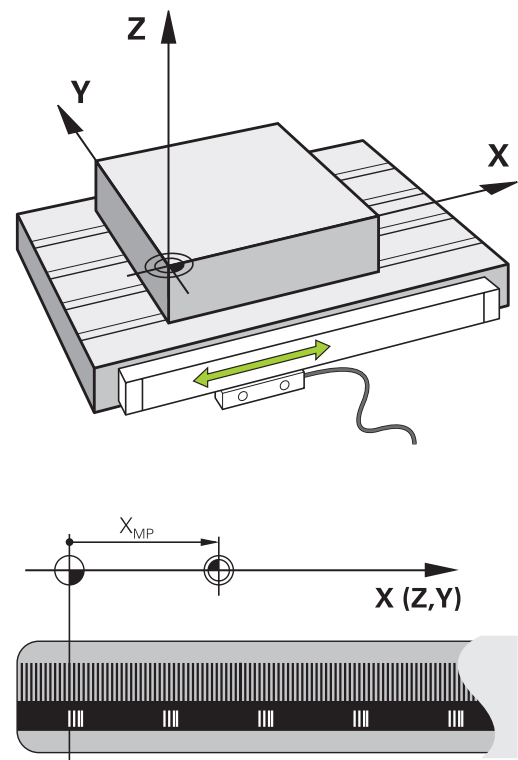
Position encoders and reference marks

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 TNC 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 association, incremental position encoders are provided with reference marks. The scales of the position encoders contain one or more reference marks that transmit a signal to the TNC when they are crossed over. From that signal the TNC can re-establish the assignment of displayed positions to machine positions. 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.

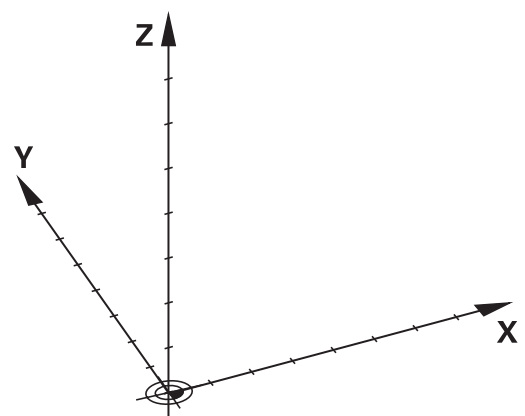


Reference system

A reference system is required to define positions in a plane or in space. The position data are always referenced to a predetermined point and are described through coordinates.

The Cartesian coordinate system (a rectangular coordinate system) is based on the three coordinate axes X, Y and Z. The axes are mutually perpendicular and intersect at one point called the datum. A coordinate identifies the distance from the datum in one of these directions. A position in a plane is thus described through two coordinates, and a position in space through three coordinates.

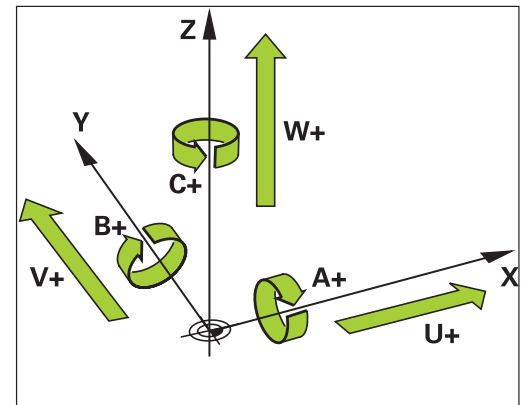
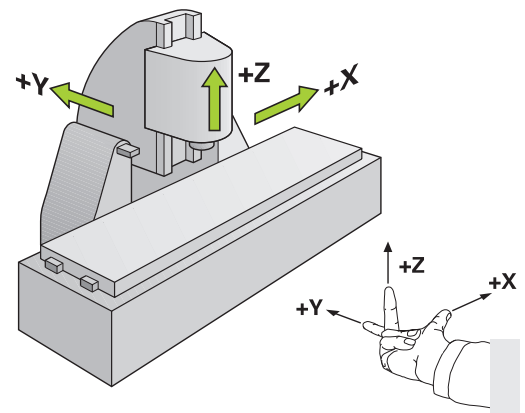
Coordinates that are referenced to the datum are referred to as absolute coordinates. Relative coordinates are referenced to any other known position (reference point) you define within the coordinate system. Relative coordinate values are also referred to as incremental coordinate values.



Reference system on milling machines

When using a milling machine, you orient tool movements to the Cartesian coordinate system. The illustration at right shows how the Cartesian coordinate system describes the machine axes. The figure illustrates the right-hand rule for remembering the three axis directions: the middle finger points in the positive direction of the tool axis from the workpiece toward the tool (the Z axis), the thumb points in the positive X direction, and the index finger in the positive Y direction.

The TNC 640 can control up to 18 axes. The axes U, V and W are secondary linear axes parallel to the main axes X, Y and Z, respectively. Rotary axes are designated as A, B and C. The illustration at lower right shows the assignment of secondary axes and rotary axes to the main axes.



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

3.1 Fundamentals

Polar coordinates

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

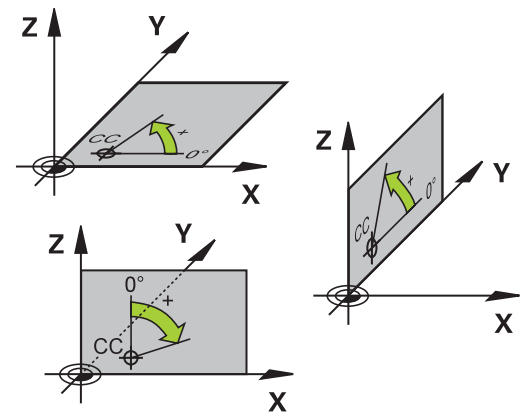
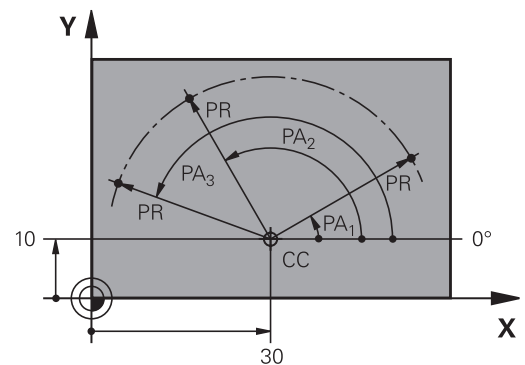
While the Cartesian coordinates X, Y and Z are three-dimensional and can describe points in space, polar coordinates are two-dimensional and describe points in a plane. Polar coordinates have their datum at a circle center (CC), or pole. A position in a plane can be clearly defined by the:

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

Setting the pole and the angle reference axis

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

Coordinates of the pole (plane)	Reference axis of the angle
X/Y	+X
Y/Z	+Y
Z/X	+Z



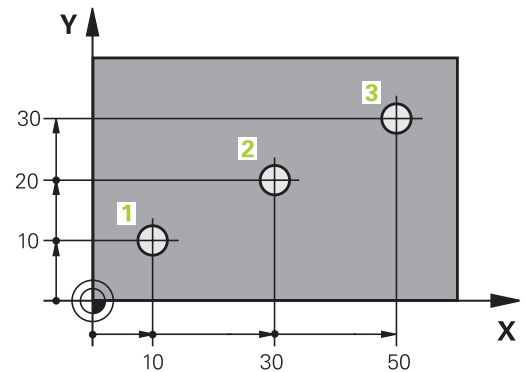
Absolute and incremental workpiece positions

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 uniquely 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 function "I" before the axis.

Example 2: Holes dimensioned in incremental coordinates

Absolute coordinates of hole 4

X = 10 mm

Y = 10 mm

Hole 5, with respect to 4

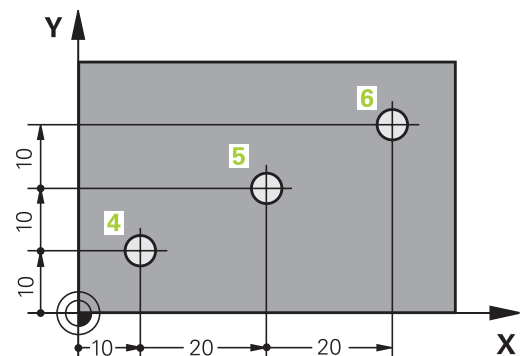
X = 20 mm

Y = 10 mm

Hole 6, with respect to 5

X = 20 mm

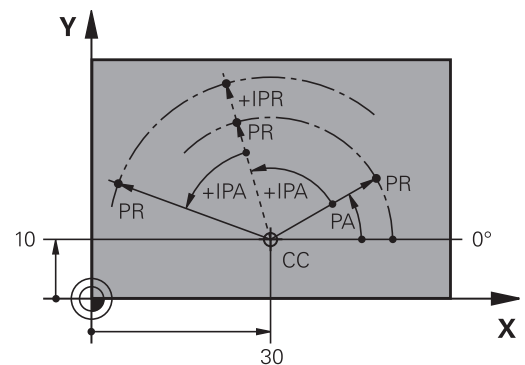
Y = 10 mm



Absolute and incremental polar coordinates

Absolute polar 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.



3.1 Fundamentals

Selecting the datum

A production drawing identifies a certain form element of the workpiece, usually a corner, as the absolute datum. When setting the datum, 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 TNC either to zero or to a known position value for each position. This establishes the reference system for the workpiece, which will be used for the TNC display and your part program.

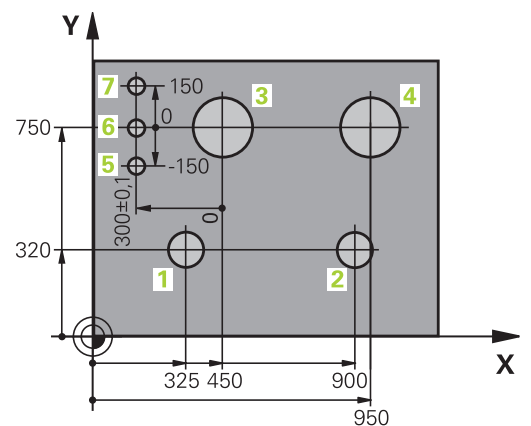
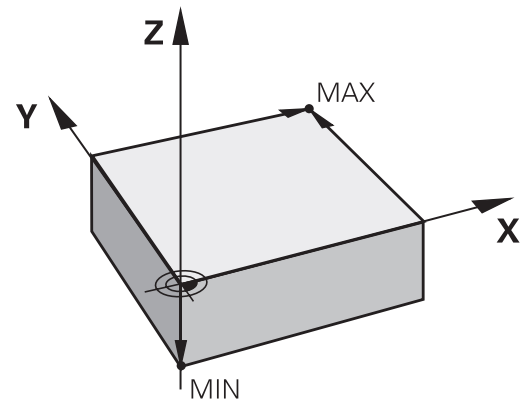
If the production drawing is dimensioned in relative coordinates, simply use the coordinate transformation cycles (see User's Manual for Cycles, Cycles for Coordinate Transformation).

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

The fastest, easiest and most accurate way of setting the datum is by using a 3-D touch probe from HEIDENHAIN. See "Setting the Datum with a 3-D Touch Probe" in the Cycle Programming User's Manual.

Example

The workpiece drawing shows holes (1 to 4) whose dimensions are shown with respect to an absolute datum with the coordinates $X=0$ $Y=0$. Holes 5 to 7 are dimensioned with respect to a relative datum with the absolute coordinates $X=450$, $Y=750$. With the **DATUM SHIFT** cycle you can temporarily set the datum to the position $X=450$, $Y=750$, to be able to program holes 5 to 7 without further calculations.



3.2 Opening programs and entering

Organization of an NC program in HEIDENHAIN Conversational format

A part program consists of a series of program blocks. The figure at right illustrates the elements of a block.

The TNC numbers the blocks of a part program in ascending sequence.

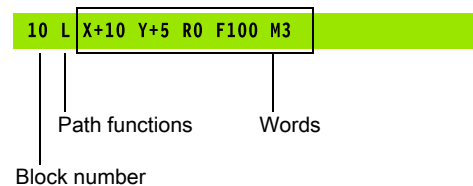
The first block of a program is identified by **BEGIN PGM**, the program name and the active unit of measure.

The subsequent blocks contain information on:

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

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

Block



After each tool call, HEIDENHAIN recommends always traversing to a safe position from which the TNC can position the tool for machining without causing a collision!

3.2 Opening programs and entering

Define the blank: BLK FORM

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



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

The TNC can depict various types of blank forms.

Soft key

Function



Define a rectangular blank



Define a cylindrical blank



Define a rotationally symmetric blank

Rectangular blank

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

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

Example: Display the BLK FORM in the NC program

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

Cylindrical blank

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

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



The **DIST** and **RI** parameters are optional and do not need to be programmed.

Example: Display the BLK FORM CYLINDER in the NC program

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

Rotationally symmetric blank of any shape

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

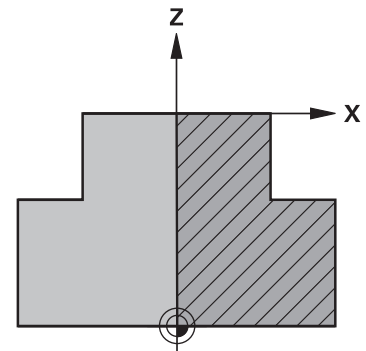
In the workpiece blank definition you refer to the contour description:

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

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



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



Programming: Fundamentals, File Management

3.2 Opening programs and entering

Example: Display the BLK FORM ROTATION in the NC program

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

Opening a new part program

You always enter a part program in the **Programming** mode of operation. An example of program initiation:

- ▶ Select the **Programming** mode of operation
- ▶ To call the file manager, press the **PGM MGT** key.

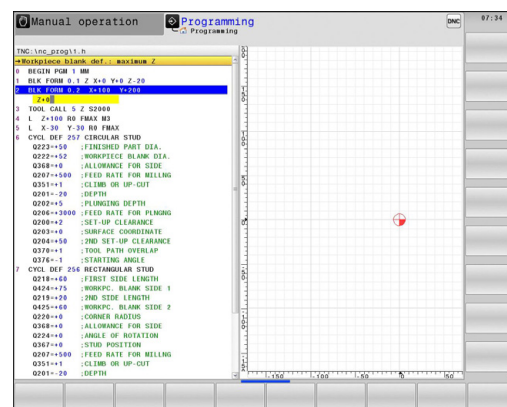
Select the directory in which you wish to store the new program:

FILE NAME = NEW.H

- ▶ Enter the new program name and confirm your entry with the ENT key.
- ▶ Select the unit of measure: Press the MM or INCH soft key. The TNC 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

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



WORKPIECE BLANK DEF.: MINIMUM

ENT

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

WORKPIECE BLANK DEF.: MAXIMUM

ENT

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

Example: Display the BLK form in the NC program

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

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

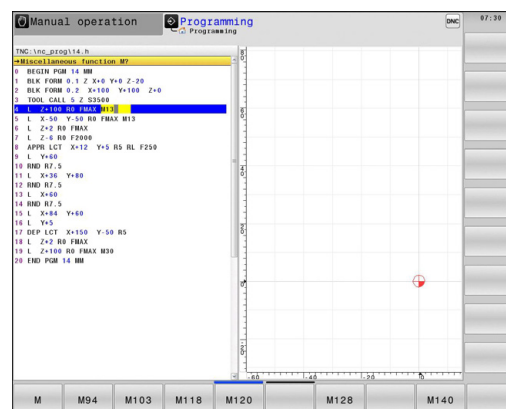


If you do not wish to define a blank form, cancel the dialog at **Working plane in graphic: XY** by pressing the DEL key.

3.2 Opening programs and entering

Programming tool movements in conversational

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



Example of a positioning block



- Open block

COORDINATES ?



- **10** (Enter the target coordinate for the X axis)



- **20** (Enter the target coordinate for the Y axis)



- go to the next question with ENT.

RADIUS COMP.: RL/RR/NO COMP.?



- Enter "No radius compensation" and go to the next question with ENT.

FEED RATE $F = ? / F_{MAX} = ENT$

- ▶ 100 (Enter a feed rate of 100 mm/min for this path contour)



- go to the next question with ENT.

MISCELLANEOUS FUNCTION M ?

- Enter **3** (miscellaneous function **M3** "Spindle ON").











- ▶ With the END key, the TNC ends this dialog.

The program-block window displays the following line:

3 L X+10 Y+5 R0 F100 M3

Possible feed rate input

Soft key	Functions for setting the feed rate
	Rapid traverse, non-modal. Exception: If defined before an APPR block, FMAX is also in effect for moving to an auxiliary point (see "Important positions for approach and departure", page 215)
	Traverse feed rate automatically calculated in TOOL CALL
	Move at the programmed feed rate (unit of measure is mm/min or 1/10 inch/min). With rotary axes, the TNC interprets the feed rate in degrees/min, regardless of whether the program is written in mm or inches
	Define the feed per revolution (units in mm/rev or inch/rev). Caution: In inch-programs, FU cannot be combined with M136
	Define the tooth feed (units in mm/tooth or inch/tooth). The number of teeth must be defined in the tool table in the CUT column.
Key	Functions for conversational guidance
	Ignore the dialog question
	End the dialog immediately
	Abort the dialog and erase the block

3.2 Opening programs and entering

Actual position capture

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

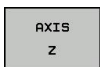
- Positioning-block programming
- Cycle programming

To transfer the correct position values, proceed as follows:

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



- ▶ Select the actual-position-capture function: In the soft-key row the TNC displays the axes whose positions can be transferred



- ▶ Select the axis: The TNC writes the current position of the selected axis into the active input box



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

In the tool axis the TNC always captures the coordinates of the tool tip and thus always takes the active tool length compensation into account.

The TNC keeps the soft-key row for axis selection active until you deactivate it by pressing the actual-position-capture key again. This behavior remains in effect even if you save the current block and open a new one with a path function key. If you select a block element in which you must choose an input alternative via soft key (e.g. for radius compensation), then the TNC also closes the soft-key row for axis selection.








The actual-position-capture function is not allowed if the tilted working plane function is active.

Editing a program







You cannot edit a program while it is being run by the TNC in a machine operating mode.

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

Soft key/ Keys	Function
	Change the position of the current block on the screen. Press this soft key to display additional program blocks that are programmed before the current block
	Change the position of the current block on the screen. Press this soft key to display additional program blocks that are programmed after the current block
	Move from one block to the next
	
	Select individual words in a block
	
	To select a certain block, press the GOTO key, enter the desired block number, and confirm with the ENT key. Or: Press the GOTO key, enter the block number step and jump up or down the number of entered lines by pressing the N LINES soft key

Programming: Fundamentals, File Management

3.2 Opening programs and entering

Soft key/key	Function
	<ul style="list-style-type: none"> ■ Set the selected word to zero ■ Erase an incorrect number ■ Delete the (clearable) error message
	Delete the selected word
	<ul style="list-style-type: none"> ■ Delete the selected block ■ Erase cycles and program sections
	Insert the block that you last edited or deleted

Inserting blocks at any desired location

- ▶ Select the block after which you want to insert a new block and initiate the dialog



Editing and inserting words

- ▶ Select a word in a block and overwrite it with the new one. The plain-language 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 blocks

Set the AUTO DRAW soft key to OFF.

-  ▶ Select a word in a block: Press the arrow key repeatedly until the highlight is on the desired word
-  ▶ Select a block with the arrow keys

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

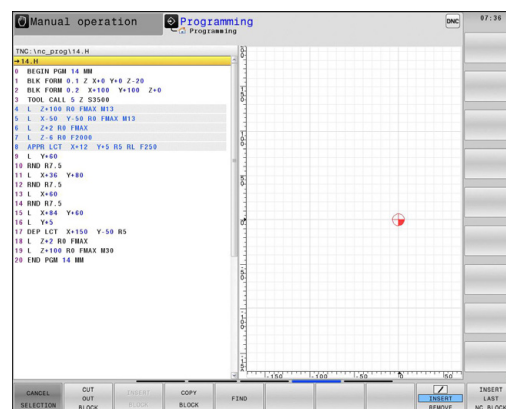


If you have started a search in a very long program, the TNC shows a progress display window. You then have the option of canceling the search via soft key.

Marking, copying, cutting and inserting program sections

The TNC 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 block of the section you wish to copy
- ▶ Mark the first block: Press the **SELECT BLOCK** soft key. The TNC then highlights the block and displays the **CANCEL SELECTION** soft key
- ▶ Move the highlight to the last block of the program section you wish to copy or cut. The TNC shows the marked 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 BLOCK** soft key. The TNC stores the selected block
- ▶ Using the arrow keys, select the block after which you wish to insert the copied (cut) program section



To insert the section into another program, select the corresponding program using the file manager and then mark the block after which you wish to insert the program section.

- ▶ Insert the saved program section: Press the **INSERT BLOCK** soft key
- ▶ To end the marking function, press the **CANCEL SELECTION** soft key

3.2 Opening programs and entering

The TNC search function

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

Finding any text

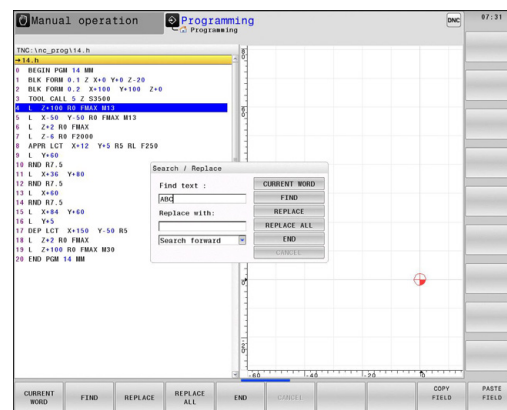
FIND

- Select the Search function: The TNC superimposes the search window and displays the available search functions in the soft-key row
- **TOOL** (Enter the text to be searched for)
- Start the search process: The TNC moves to the next block containing the text you are searching for
- Repeat the search process: The TNC moves to the next block containing the text you are searching for
- End the search function

FIND

FIND

END



Finding/Replacing any text



The find/replace function is not possible if

- a program is protected
- the program is currently being run by the TNC

When using the **REPLACE ALL** function, ensure that you do not accidentally replace text that you do not want to change. Once replaced, such text cannot be restored.

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

FIND

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

FIND

- ▶ Start the search process: The TNC 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

- ▶ End the search function

3.3 File Management: Fundamentals

Files

Files in the TNC	Type
Programs	
in HEIDENHAIN format	.H
in DIN/ISO format	.I
Compatible 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 files (e.g. structure items)	.DEP
Freely definable tables	.TAB
Pallets	.P
Turning tools	.TRN
Texts as	
ASCII files	.A
Protocol files	.TXT
Help files	.CHM
CAD files as	
ASCII files	DXF .IGES .STEP

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

The TNC 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 TNC. 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 TNC generates a backup file (*.bak) after editing and saving of NC programs. This can reduce the memory space available to you.

File names

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

File name	File type
PROG20	.H

File names should not exceed 24 characters, otherwise the TNC cannot display the entire file name.

File names on the TNC must comply with this standard: The Open Group Base Specifications Issue 6 IEEE Std 1003.1, 2004 Edition (Posix-Standard). Accordingly, the file names may include the characters below:

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 . _ -

You should not use any other characters in file names in order to prevent any file transfer problems.



The maximum limit for the path and file name together is 255 characters, see "Paths", page 117.

3.3 File Management: Fundamentals

Displaying externally generated files on the TNC

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

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

For further information about displaying and editing the listed file types: see page 129

Data Backup

We recommend saving newly written programs and files on a PC at regular intervals.

The TNCremo data transmission freeware from HEIDENHAIN is a simple and convenient method for backing up data stored on the TNC.

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



Saving the contents of the internal memory can take up to several hours. In this case, it is a good idea to save the data outside of work hours, e.g. during the night.

Take the time occasionally to delete any unneeded files so that the TNC always has enough memory space for system files (such as the tool table).



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

3.4 Working with the File Manager

Directories

To ensure that you can easily find your files, we recommend that you organize your internal memory into directories. 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 "\".



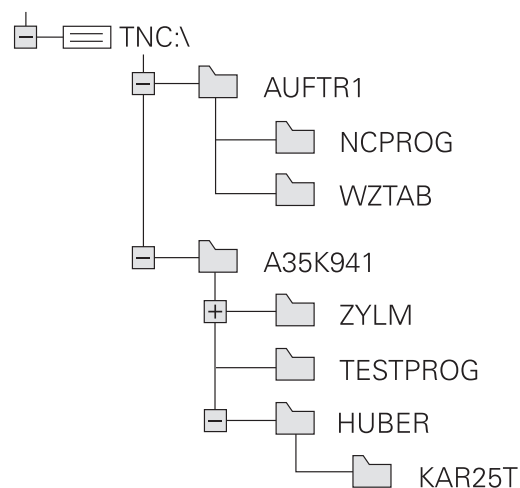
The path, including all drive characters, directory and the file name, including the extension, must not exceed 255 characters!

Example

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

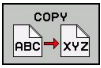





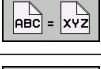


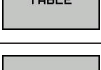

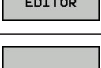
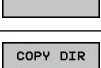
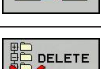




TNC:\AUFTR1\NCPRG\PROG1.H

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



3.4 Working with the File Manager

Overview: Functions of the file manager

Soft key	Function	Page
	Copy a single file	121
	Display a specific file type	120
	Create new file	121
	Display the last 10 files that were selected	124
	Delete a file	125
	Tag a file	126
	Rename a file	127
	Protect a file against editing and erasure	128
	Cancel file protection	128
	Import a tool table	192
	Manage network drives	138
	Select the editor	128
	Sort files by properties	127
	Copy a directory	123
	Delete directory with all its subdirectories	
	Display all the directories of a particular drive	
	Rename a directory	
	Create a new directory	

Calling the File Manager



- Press the PGM MGT key: The TNC displays the file management window (see figure for default setting). If the TNC 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. One drive is the internal memory of the TNC. Other drives are the interfaces (RS232, Ethernet), which can be used, for example, for connecting a personal computer. 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.

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.

The screenshot shows the 'Manual operation' screen with the 'Programming' window active. On the left, a directory tree lists drives like 'P1', 'P2', 'P3', 'P4', 'P5', 'P6', 'P7', 'P8', 'P9', 'P10', 'P11', 'P12', 'P13', 'P14', 'P15', 'P16', 'P17', 'P18', 'P19', 'P20', 'P21', 'P22', 'P23', 'P24', 'P25', 'P26', 'P27', 'P28', 'P29', 'P30', 'P31', 'P32', 'P33', 'P34', 'P35', 'P36', 'P37', 'P38', 'P39', 'P40', 'P41', 'P42', 'P43', 'P44', 'P45', 'P46', 'P47', 'P48', 'P49', 'P50', 'P51', 'P52', 'P53', 'P54', 'P55', 'P56', 'P57', 'P58', 'P59', 'P60', 'P61', 'P62', 'P63', 'P64', 'P65', 'P66', 'P67', 'P68', 'P69', 'P70', 'P71', 'P72', 'P73', 'P74', 'P75', 'P76', 'P77', 'P78', 'P79', 'P80', 'P81', 'P82', 'P83', 'P84', 'P85', 'P86', 'P87', 'P88', 'P89', 'P90', 'P91', 'P92', 'P93', 'P94', 'P95', 'P96', 'P97', 'P98', 'P99', 'P100'. On the right, a table lists files with columns: File name, Byte, Status, Date, Time. The file 'P1.H' is selected.

File name	Byte	Status	Date	Time
P1.H	112		17-12-2013	07:33:01
P2.H	2087		18-12-2013	13:18:52
P3.H	6879		18-12-2013	13:18:52
P4.H	481		18-12-2013	14:01:14
P5.H	4827		18-12-2013	13:18:52
P6.H	6398		18-12-2013	13:18:52
P7.H	289		18-12-2013	13:18:52
P8.H	3599		18-12-2013	13:18:52
P9.H	479		18-12-2013	13:18:52
P10.H	623		18-12-2013	13:18:52
P11.H	1223		17-12-2013	07:31:35
P12.H	2065		18-12-2013	13:18:52
P13.H	1292		17-12-2013	07:22:28
P14.H	11195		18-12-2013	13:18:52
P15.H	126718		18-12-2013	13:18:54
P16.H	6557		18-12-2013	13:18:54

Display	Meaning
File name	File name (max. 25 characters) and file type
Byte	File size in bytes
Status	File properties:
E	Program is selected in the Programming mode of operation
S	Program is selected in the Test Run mode of operation
M	Program is selected in a Program Run mode of operation
+	Program has dependent files with the DEP extension that are not displayed, 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 **CfgPgmMgt/dependentFiles** to **MANUAL**.

3.4 Working with the File Manager

Selecting drives, directories and files



- ▶ Calling the File Manager

Use the arrow keys or the soft keys to move the highlight to the desired position on the screen:



- ▶ Moves the highlight from the left to the right window, and vice versa



- ▶ Moves the highlight up and down within a window



- ▶ Moves the highlight one page up or down within a window



Step 1: Select drive

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



- ▶ To select a drive, press the SELECT soft key, or



- ▶ 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



- ▶ To display all files, press the SHOW ALL soft key or

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



- ▶ Press the SELECT soft key, or



- ▶ Press the ENT key

The TNC opens the selected file in the operating mode from which you called the file manager

Creating a new directory

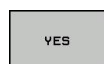
- ▶ Move the highlight in the left window to the directory in which you want to create a subdirectory



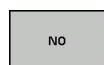
- ▶ Press the **NEW DIRECTORY** soft key
- ▶ Enter a directory name
- ▶ Press the **ENT** key



DIRECTORY \CREATE NEW ?



- ▶ Press the **YES** soft key to confirm, or



- ▶ the **NO** soft key to abort.

Creating a new file

- ▶ Select the directory in the left window in which you wish to create the new file
- ▶ Position the cursor in the right window

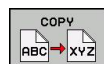


- ▶ Press the **NEW FILE** soft key
- ▶ Enter the file name with file extension
- ▶ Press the **ENT** key

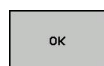


Copying a single file

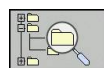
- ▶ Move the highlight to the file you wish to copy



- ▶ Press the **COPY** soft key: Select the copying function. The TNC opens a pop-up window



- ▶ Enter the name of the destination file and confirm your entry with the **ENT** key or the **OK** soft key: The TNC copies the file into the active directory or into the selected target directory. The original file is retained, or



- ▶ Press the **Target Directory** soft key to call a pop-up window in which you select the target directory by pressing the **ENT** key or the **OK** soft key: the TNC copies the file to the selected directory. The original file is retained.



When the copying process has been started with **ENT** or the **OK** soft key, the TNC displays a pop-up window with a progress indicator.

3.4 Working with the File Manager

Copying files into another directory

- ▶ Select a screen layout with two equally sized windows
- ▶ To display directories in both windows, press the **PATH** soft key

In the right window

- ▶ Move the highlight 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

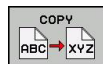
- ▶ Select the directory with the files that you wish to copy and press the **ENT** key to display the files in this directory



- ▶ Call the file tagging functions



- ▶ Move the highlight to the file you want to copy and tag it. You can tag several files in this way, if desired



- ▶ Copy the tagged files into the target directory

Further tagging functions: see "Tagging files", page 126.

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

Overwriting files

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

- ▶ To overwrite all files ("Existing files" check box selected), press the OK soft key, or
- ▶ Press the CANCEL soft key if no file is to be overwritten

If you wish to overwrite a protected file, you need to select the "Protected files" check box or cancel the copying 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 already exist
- The file to be copied must only contain the lines you want to replace
- Both tables must have the same file extension



The **REPLACE FIELDS** function is used to overwrite lines in the target table. To avoid losing data, create a backup copy of the original table.

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).

- ▶ Copy this table from the external data medium to any directory
- ▶ Copy the externally created table to the existing table TOOL.T using the TNC file manager. The TNC asks if you wish to overwrite the existing TOOL.T tool table:
- ▶ If you press the **YES** soft key, the TNC will completely overwrite the current TOOL.T tool table. After the copying process the new TOOL.T table consists of 10 lines.
- ▶ Or press the **REPLACE FIELDS** soft key for the TNC to overwrite 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.

- ▶ 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 TNC opens the window for selecting the target directory
- ▶ Select the target directory and confirm with **ENT** or the **OK** soft key: The TNC copies the selected directory and all its subdirectories to the selected target directory

3.4 Working with the File Manager

Choosing one of the last files selected



- ▶ Calling the File Manager

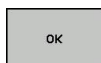


- ▶ To display the 10 files last selected: Press the **LAST FILES** soft key.

Use the arrow keys to move the highlight to the file you wish to select:



- ▶ Moves the highlight up and down within a window



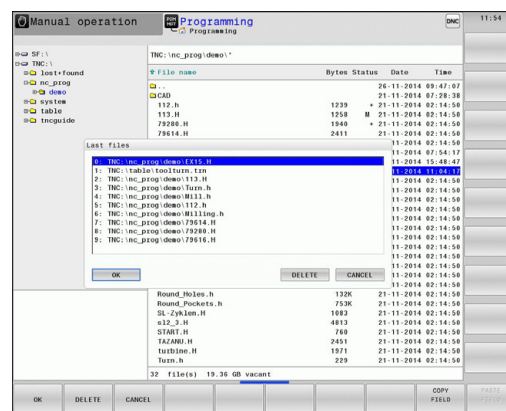
- ▶ To select a file: Press the **OK** soft key, or...



- ▶ Press the **ENT** key



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



Deleting a file



Caution: Data may be lost!

Once you delete files they cannot be restored!

- Move the highlight to the file you want to delete



- To select the erasing function: Press the **DELETE** soft key. The TNC asks whether you really want to delete the file
- To confirm deletion: press the **OK** soft key, or
- To interrupt deletion: Press the **CANCEL** soft key

Deleting a directory



Caution: Data may be lost!

Once you delete files they cannot be restored!






- Move the highlight to the directory you want to delete



- To select the erasing function, press the **DELETE** soft key. The TNC inquires whether you really intend to delete the directory and all its subdirectories and files
- To confirm the deletion, press the **OK** soft key, or
- To cancel deletion, press the **CANCEL** soft key

3.4 Working with the File Manager

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 highlight to the first file



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



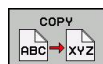
- Tag a file by pressing the TAG FILE soft key.



- Move the highlight to the next file you wish to tag: Only works via soft keys. Do not use the arrow keys!



- To tag further files, press the **TAG FILES** soft key, etc.



- Copy the tagged files: Press the **COPY** soft key, or

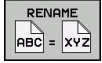


- Delete the tagged files: Leave the active soft key and then press the **DELETE** soft key to delete the tagged files



Renaming a file

- ▶ Move the highlight to the file you wish to rename



- ▶ Select the renaming function
- ▶ 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



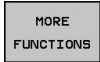
- ▶ Select the SORT soft key
- ▶ Select the soft key with the corresponding display criterion

3.4 Working with the File Manager

Additional functions

Protecting a file / Canceling file protection

- ▶ Move the highlight to the file you want to protect



- ▶ Select the miscellaneous functions: press the **soft** key



- ▶ Enable file protection: Press the **soft** key. The file is tagged with the "protected" symbol



- ▶ To cancel file protection, press the **soft** key

Selecting the editor

- ▶ Move the highlight in the right window onto the file you want to open



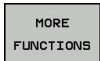
- ▶ Select the miscellaneous functions: press the **soft** key



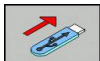
- ▶ To select the editor with which to open the selected file, press the **soft** key
- ▶ Mark the desired editor
- ▶ Press the **OK** soft key to open the file

Connecting/removing a USB device

- ▶ Move the highlight to the left window



- ▶ Select the miscellaneous functions: press the **soft** key



- ▶ Shift the soft-key row
- ▶ Search for a USB device

- ▶ To remove the USB device, move the highlight to the USB device in the directory tree



- ▶ Remove the USB device

For more information: see "USB devices on the TNC", page 139.

Additional tools for management of external file types

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

File types	Description
PDF files (pdf)	page 129
Excel spreadsheets (xls, csv)	page 131
Internet files (htm, html)	page 132
ZIP archives (zip)	page 133
Text files (ASCII files, e.g. txt, ini)	page 134
Video files	page 134
Graphics files (bmp, jpg, gif, png)	page 135



If you transfer files from a PC to the control by means of TNCremo, you must have entered the file name extension pdf, xls, zip, bmp gif, jpg and png in the list of the file types for binary transmission (menu item **Extras > Configuration > Mode** in TNCremo).

Displaying PDF files

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

PGM
MGT

- ▶ Calling the File Manager
- ▶ Select the directory in which the PDF file is saved
- ▶ Move the highlight to the PDF file
- ▶ Press ENT: The TNC opens the PDF file in its own application using the **PDF viewer** additional tool

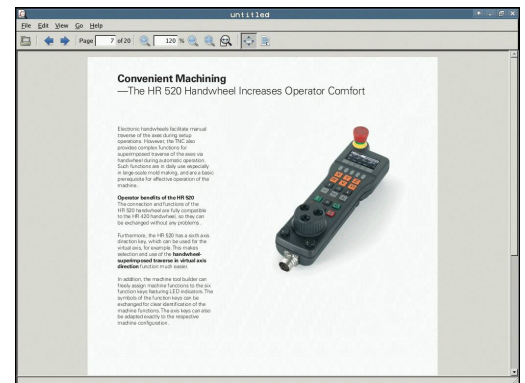
ENT



With the key combination ALT+TAB you can always return to the TNC user interface while leaving the PDF file open. Alternatively, you can also click the corresponding symbol in the task bar to switch back to the TNC interface.



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



3.4 Working with the File Manager

To exit the **PDF viewer**, proceed as follows:

- ▶ Use the mouse to select the **File** menu item
- ▶ Select the menu item **Close**: The TNC returns to the file manager

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



- ▶ Press the key for switching the soft keys: The **PDF viewer** opens the **File** pull-down menu



- ▶ Select the **Close** menu item and confirm with the **ENT** key: The TNC returns to the file manager



Displaying and editing Excel files

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



- ▶ Calling the File Manager
- ▶ Select the directory in which the Excel file is saved
- ▶ Move the highlight to the Excel file
- ▶ Press ENT: The TNC opens the Excel file in its own application using the **Gnumeric** additional tool



With the key combination ALT+TAB you can always return to the TNC user interface while leaving the Excel file open. Alternatively, you can also click the corresponding symbol in the task bar to switch back to the TNC interface.



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

To exit **Gnumeric**, proceed as follows:

- ▶ Use the mouse to select the **File** menu item
- ▶ Select the menu item **Close**: The TNC returns to the file manager

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



- ▶ Press the key for switching the soft keys: The **Gnumeric** additional tool opens the **File** pull-down menu



- ▶ Select the **Close** menu item and confirm with the **ENT** key: The TNC returns to the file manager



Programming: Fundamentals, File Management

3.4 Working with the File Manager

Displaying Internet files

To open Internet files with the extension **htm** or **html** directly on the TNC, proceed as follows:



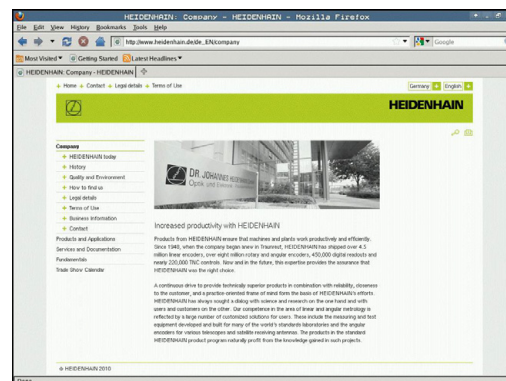
- ▶ Calling the File Manager
- ▶ Select the directory in which the Internet file is saved
- ▶ Move the highlight to the Internet file
- ▶ Press ENT: The TNC opens the Internet file in its own application using the **Mozilla Firefox** additional tool



With the key combination ALT+TAB you can always return to the TNC user interface while leaving the PDF file open. Alternatively, you can also click the corresponding symbol in the task bar to switch back to the TNC interface.



If you position the mouse pointer over a button, a brief tooltip explaining the function of this button will be displayed. More information on how to use **Mozilla Firefox** is provided under **Help**.



To exit **Mozilla Firefox**, proceed as follows:

- ▶ Use the mouse to select the **File** menu item
- ▶ Select the menu item **Quit**: The TNC returns to the file manager

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



- ▶ Press the key for switching the soft keys: The **Mozilla Firefox** opens the **File** pull-down menu



- ▶ Select the **Quit** menu item and confirm with the **ENT** key: The TNC returns to the file manager



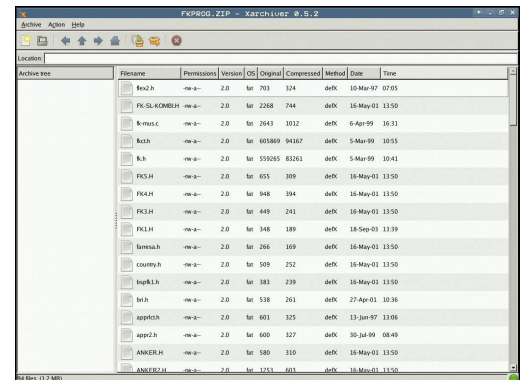
Working with ZIP archives

To open ZIP archives with the extension **zip** directly on the TNC, proceed as follows:

PGM
MGT

- ▶ Calling the File Manager
- ▶ Select the directory in which the archive file is saved
- ▶ Move the highlight to the archive file
- ▶ Press ENT: The TNC opens the archive file in its own application using the **Xarchiver** additional tool

ENT



With the key combination ALT+TAB you can always return to the TNC user interface while leaving the archive file open. Alternatively, you can also click the corresponding symbol in the task bar to switch back to the TNC interface.



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



Please note that the TNC does not carry out any binary-to-ASCII conversion or vice versa when compressing or decompressing NC programs and NC tables. When such files are transferred to TNC controls using other software versions, the TNC may not be able to read them.

To exit **Xarchiver**, proceed as follows:

- ▶ Use the mouse to select the **Archive** menu item
- ▶ Select the menu item **Quit**: The TNC returns to the file manager

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

▶

- ▶ Press the key for switching the soft keys: The **Xarchiver** opens the **Archive** pull-down menu

↓

- ▶ Select the **Quit** menu item and confirm with the **ENT** key: The TNC returns to the file manager

ENT

Programming: Fundamentals, File Management

3.4 Working with the File Manager

Displaying and editing text files

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

PGM
MGT

- ▶ Calling the File Manager
- ▶ Select the drive and the directory in which the text file is saved
- ▶ Move the highlight to the text file
- ▶ Press the ENT key: The TNC opens the text file with the internal text editor

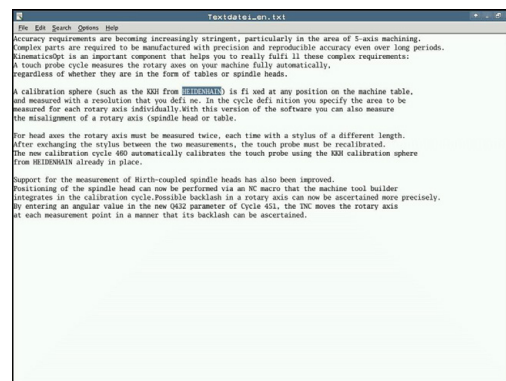
ENT



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



With the key combination ALT+TAB you can always return to the TNC user interface while leaving the text file open. Alternatively, you can also click the corresponding symbol in the task bar to switch back to the TNC interface.



To open **Leafpad**, proceed as follows:

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

To exit **Leafpad**, proceed as follows:

- ▶ Use the mouse to select the **File** menu item
- ▶ Select the menu item **Quit**: The TNC returns to the file manager

Displaying video files



This feature must be enabled and adapted by the machine tool builder.
Refer to your machine manual.

To open video files directly on the TNC, proceed as follows:

PGM
MGT

- ▶ Call the File Manager
- ▶ Select the directory in which the video file is saved
- ▶ Move the highlight to the video file
- ▶ Press ENT: The TNC opens the video file in its own application

ENT

Displaying graphic files

To open graphics files with the extension bmp, gif, jpg or png directly on the TNC, proceed as follows:

- ▶ Call the File Manager
- ▶ Select the directory in which the graphics file is saved
- ▶ Move the highlight to the graphics file
- ▶ Press ENT: The TNC opens the graphics file in its own application using the **ristretto** additional tool



With the key combination ALT+TAB you can always return to the TNC user interface while leaving the graphics file open. Alternatively, you can also click the corresponding symbol in the task bar to switch back to the TNC interface.



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



To exit **ristretto**, proceed as follows:

- ▶ Use the mouse to select the **File** menu item
- ▶ Select the menu item **Quit**: The TNC returns to the file manager

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

- ▶ Press the key for switching the soft keys: The **ristretto** additional tool opens the **File** pull-down menu
- ▶ Select the **Quit** menu item and confirm with the **ENT** key: The TNC returns to the file manager



ENT

Data transfer to/from an external data medium



Before you can transfer data to an external data medium, you must set up the data interface (see "Setting up data interfaces", page 624).

Depending on the data transfer software you use, problems can occur occasionally when you transmit data over a serial interface. They can be overcome by repeating the transmission.

PGM
MGT

- Call the File Manager



- Select the screen layout for data transfer: Press the WINDOW soft key.

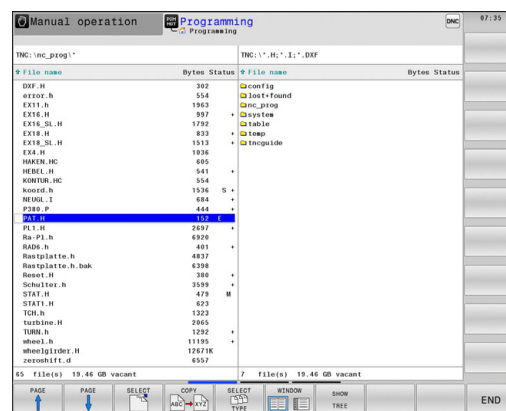
Use the arrow keys to highlight the file(s) that you want to transfer:



- Moves the highlight up and down within a window



- Moves the highlight from the right to the left window, and vice versa



If you wish to copy from the TNC to the external data medium, move the highlight in the left window to the file to be transferred.

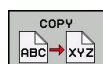
If you wish to copy from the external data medium to the TNC, move the highlight in the right window to the file to be transferred.



- ▶ Select another drive or directory: Press the **SHOW TREE** soft key



- ▶ Use the arrow keys to select the desired directory
- ▶ Select the desired file: Press the **SHOW FILES** soft key



- ▶ Use the arrow keys to select the file
- ▶ Transfer a single file: Press the **COPY** soft key

- ▶ Confirm with the **OK** soft key or with the **ENT** key. A status window appears on the TNC, informing about the copying progress, or



- ▶ Stop transfer: Press the **WINDOW** soft key. The TNC displays the standard file manager window again

3.4 Working with the File Manager

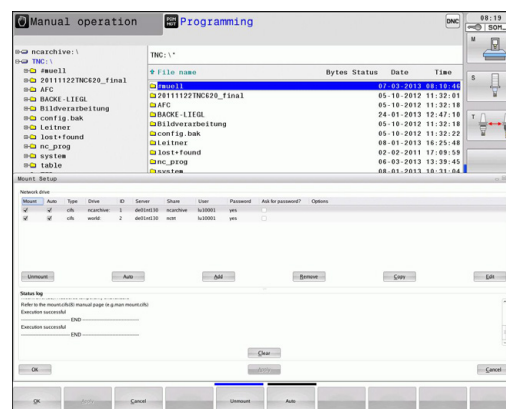
The TNC in a network



To connect the Ethernet card to your network, see "Ethernet interface", page 631.

The TNC logs error messages during network operation, see "Ethernet interface", page 631.

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



Connecting and disconnecting a network drive

PGM MGT

- ▶ To select the program management: Press the **PGM MGT** key. If necessary, press the **WINDOW** soft key to set up the screen as it is shown at the upper right

NET

- ▶ To select the network settings: Press the **NETWORK** soft key (second soft key row).
- ▶ To manage the network drives: Press the **DEFINE NETWORK CONNECTN.** soft key. In a window the TNC shows the network drives available for access. The soft keys described below are used to define the connection for each drive

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

USB devices on the TNC



Caution: Data may be lost!

Only use the USB interface for transferring and saving, not for processing or running programs.

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

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

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



If an error message is displayed when connecting a USB data medium, check the setting in the SELinux security software. ("SELinux security software", page 92)

The TNC also displays the **USB: TNC does not support device** error message if you connect a USB hub. In this case, simply acknowledge the message with the CE key.

In theory, you should be able to connect all USB devices with the file systems mentioned above to the TNC. It may happen that a USB device is not identified correctly by the control. In such cases, use another USB device.

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











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


3.4 Working with the File Manager

Remove the USB device

To remove a USB device, proceed as follows:

- 
 - ▶ To call the file manager, press the **PGM MGT** key
- 
 - ▶ Select the left window with the arrow key
- 
 - ▶ Use the arrow keys to select the USB device to be removed
- 
 - ▶ Scroll through the soft-key row
- 
 - ▶ Select additional functions
- 
 - ▶ Scroll through the soft-key row
- 
 - ▶ Select the function for removing USB devices. The TNC removes the USB device from the directory tree and reports **The USB device can be removed now.**
- 
 - ▶ Remove the USB device
 - ▶ Quit the File Manager

In order to re-establish a connection with a USB device that has been removed, press the following soft key:

- 
 - ▶ Select the function for reconnection of USB devices

4

**Programming:
Programming aids**

4.1 Adding comments

4.1 Adding comments

Application

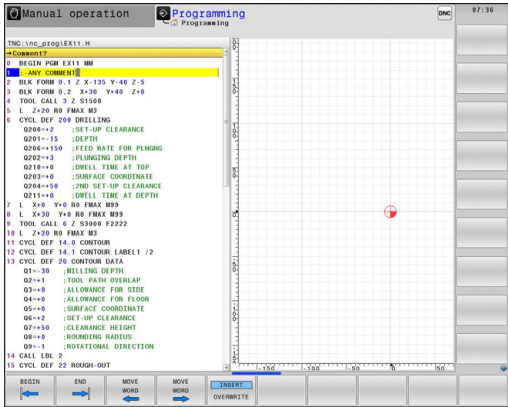
You can add comments to a part program to explain program steps or make general notes.



Depending on the **lineBreak** machine parameter, the TNC displays comments that can no longer be completely displayed on the screen, in several lines, or the >> character appears on the screen.

The last character in a comment block must not have any tilde (~).

You have the following possibilities for adding comments.



Entering comments during programming

- ▶ Enter the data for a program block, then press the semicolon key ";" on the alphabetic keyboard—the TNC displays the dialog prompt **COMMENT?**
- ▶ Enter your comment and conclude the block by pressing the **END** key





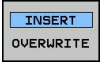
Inserting comments after program entry

- ▶ Select the block to which a comment is to be added
- ▶ Select the last word in the block with the right arrow key, then press the semicolon key (;): The TNC displays the dialog prompt **COMMENT?**
- ▶ Enter your comment and conclude the block by pressing the **END** key

Entering a comment in a separate block

- ▶ Select the block after which the comment is to be inserted
- ▶ Initiate the programming dialog with the semicolon key (;) on the alphabetic keyboard
- ▶ Enter your comment and conclude the block by pressing 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. Words must be separated by a space
	Jump to the end of a word. Words must be separated by a space
	Switch between paste and overwrite mode

4 Programming: Programming aids

4.2 Display of NC Programs

4.2 Display of NC Programs

Syntax highlighting

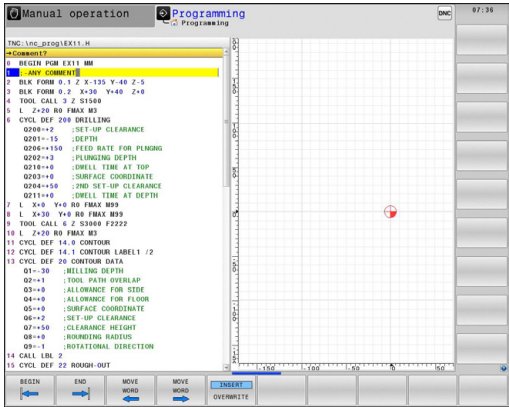
The TNC displays syntax elements with various colors according to their meaning. Programs are made more legible and clear with color-highlighting.

Color highlighting of syntax elements

Use	Color
Standard color	Black
Display of comments	Green
Display of numerical values	Blue
Block number	Purple

Scrollbar

You can move the screen content with the mouse via the scrollbar on the right edge of the program window. In addition, the size and position of the scrollbar indicates program length and cursor position.



4.3 Structuring programs

Definition and applications

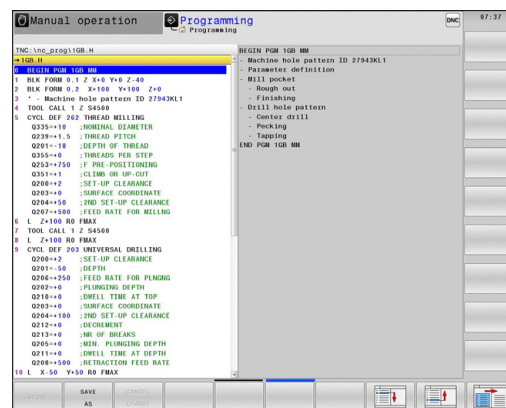
This TNC function enables you to comment part programs in structuring blocks. Structuring blocks are short texts with up to 252 characters and are used as comments or headlines for the subsequent program lines.

With the aid of appropriate structuring blocks, you can organize long and complex programs in a clear and comprehensible manner.

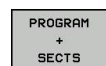
This function is particularly convenient if you want to change the program later. Structuring blocks can be inserted into the part program at any point.

They can also be displayed in a separate window. Use the appropriate screen layout for this.

The inserted structure items are managed by the TNC in a separate file (extension: .SEC.DEF). This speeds navigation in the program structure window.



Displaying the program structure window / Changing the active window



- ▶ Display the program structure window: Select the **PGM + SECTS** screen display



- ▶ Switch the active window: Press the **CHANGE WINDOW** soft key

Inserting a structuring block in the program window

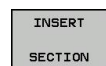
- ▶ Select the block after which the structuring block is to be inserted



- ▶ Press the **SPEC FCT** key



- ▶ Press the **PROGRAMMING AIDS** soft key



- ▶ Press the **INSERT SECTION** soft key or the * key on the ASCII keyboard



- ▶ Enter the structuring text
- ▶ If necessary, change the structure depth with the soft key

Selecting blocks in the program structure window

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

4.4 Calculator

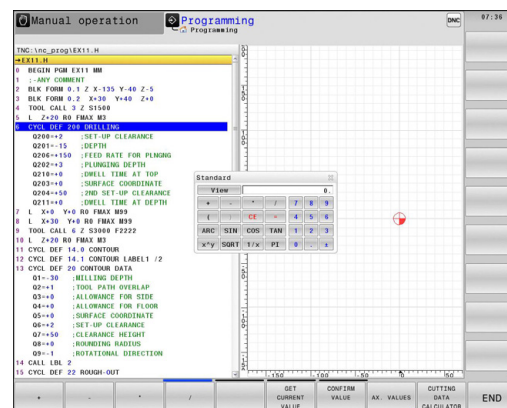
4.4 Calculator

Operation

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

- Use the **CALC** key to show and hide the on-line calculator
- Select the arithmetical functions: The calculator is operated with short commands via soft key or through the alphabetic keyboard.

Calculate function	Shortcut (soft key)
Addition	+
Subtraction	-
Multiplication	*
Division	/
Calculations in 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 integers	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 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 actual-position-capture key or the **CONFIRM VALUE** soft key for the TNC to transfer the value into the active input box and closes the calculator










You can also transfer values from a program into the calculator. When you press the GET CURRENT VALUE soft key of the GOTO key, the TNC 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.


4

Programming: Programming aids

4.4 Calculator

Functions in the pocket calculator

Soft key	Function
	Load the nominal or reference value of the respective axis position into the calculator
	Load the numerical value from the active input field into the calculator
	Load the numerical value from the calculator field into the active input field
	Copy the numerical value from the calculator
	Insert the copied numerical value into the calculator
	Open the cutting data calculator
	Position the calculator in the center



You can also shift the calculator with the arrow keys on your keyboard. If you have connected a mouse you can also position the calculator with this.

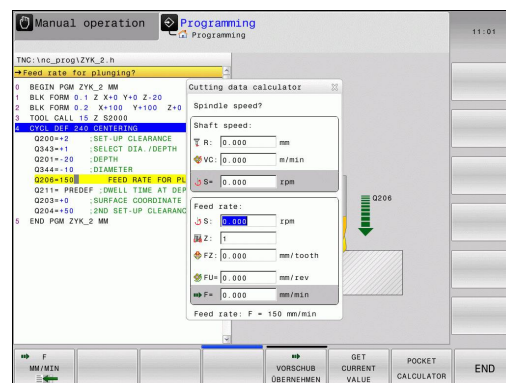
4.5 Cutting data calculator

Application

With the cutting data calculator you can calculate the spindle speed and the feed rate for a machine 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. During turning, feed rates are usually defined in mm per revolution (mm/rev) (**M136**), but the cutting data calculator always calculates in mm per minute (mm/min). In addition, the radius in the cutting data calculator is with respect to the tool, but turning operations need the tool diameter.



To open the cutting data calculator, press the **CUTTING DATA CALCULATOR** soft key. The TNC shows the soft key if you

- open the calculator (**CALC** key)
- open the dialog field for spindle speed input in the TOOL CALL block
- open the dialog field for feed rate input in positioning blocks or cycles
- enter a feed rate in manual operation (F soft key)
- enter a spindle speed in manual operation (S soft key)

The cutting data calculator is displayed with different input fields depending on whether you calculate a spindle speed or a feed rate:


Window or spindle speed calculation:

Code letter	Meaning
R:	Tool radius (mm)
VC:	Cutting speed (m/min)
S=	Result for spindle speed (rev/min)

4.5 Cutting data calculator









Window for feed rate calculation:









Code letter	Meaning
S:	Spindle speed (rpm)
Z:	Number of teeth on the tool (n)
FZ:	Feed per tooth (mm/tooth)
FU:	Feed per revolution (mm/rev)
F=	Result for feed rate (mm/min)



You can also calculate the feed rate in the TOOL CALL block and automatically transfer it to the subsequent positioning blocks and cycles. For feed rate input in positioning blocks or cycles, select the soft key F AUTO. The TNC then uses the feed rate defined in the TOOL CALL block. If you have to change the feed rate later, you only need to adjust the feed-rate value in the TOOL CALL block.

Functions in the cutting data calculator:

Soft key	Function
	Load the spindle speed from the cutting data calculator form into an open dialog field.
	Load the feed rate from the cutting data calculator form into an open dialog field.
	Load the cutting speed from the cutting data calculator form into an open dialog field.
	Load the feed per tooth from the cutting data calculator form into an open dialog field.
	Load the feed per revolution from the cutting data calculator form into an open dialog field.
	Load the tool radius into the cutting data calculator form
	Load the spindle speed from the open dialog field into the cutting data calculator form
	Load the feed rate from the open dialog field into the cutting data calculator form

Soft key	Function
	Load the feed per revolution from the open dialog field into the cutting data calculator form
	Load the feed per tooth from the open dialog field into the cutting data calculator form
	Load the value from an open dialog field into the cutting data calculator form
	Switch to the pocket calculator
	Move the cutting data calculator in the direction of the arrow
	Position the cutting data calculator in the center
	Use inch values in the cutting data calculator
	Close the cutting data calculator

4.6 Programming graphics

Generate/do not generate graphics during programming


While you are writing the part program, you can have the TNC generate a 2-D pencil-trace graphic of the programmed contour.

► Switch the screen layout to displaying program blocks to the left and graphics to the right: Press the screen layout key and the **PROGRAM + GRAPHICS** soft key



- Set the **AUTO DRAW** soft key to **ON** . While you are entering the program lines, the TNC generates each programmed path contour in the graphics window in the right screen half

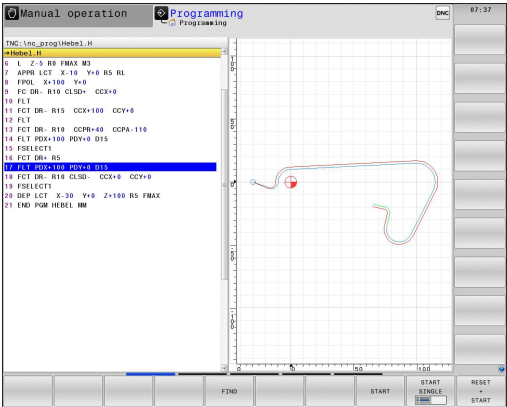
If you do not wish to have graphics generated during programming, set the **AUTO DRAW** soft key to **OFF** .



If **AUTO DRAW** is set to **ON**, during generation of the 2-D line graphic the control does not consider:

- Program section repeats
- Jump commands
- M functions, such as M2 or M30
- Cycle calls

Only use automatic drawing during contour programming.



Generating a graphic for an existing program

- Use the arrow keys to select the block up to which you want the graphic to be generated, or press **GOTO** and enter the desired block number



- To generate graphics, press the **RESET + START** soft key

Additional functions:

Soft key	Function
	Generate a complete graphic
	Generate programming graphic blockwise
	Generate a complete graphic or complete it after RESET + START
	Stop the programming graphics. This soft key only appears while the TNC is generating the programming graphics
	Select plan view
	Select front view
	Select side view

4

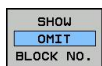
Programming: Programming aids

4.6 Programming graphics

Block number display ON/OFF



- ▶ Shift the soft-key row

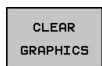


- ▶ To show block numbers: Set the **SHOW OMIT BLOCK NO.** soft key to **SHOW**
- ▶ Hide block numbers: Set the **SHOW OMIT BLOCK NO.** soft key to **OMIT**

Erasing the graphic



- ▶ Shift the soft-key row

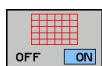


- ▶ Erase graphic: Press **CLEAR GRAPHICS** soft key

Showing grid lines



- ▶ Shift the soft-key row



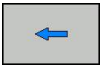



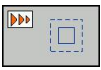
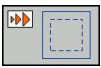
- ▶ Show grid lines: Press the **SHOW GRID LINES** soft key

Magnification or reduction of details

You can select the graphics display

- Shift the soft-key row (second row, see figure)

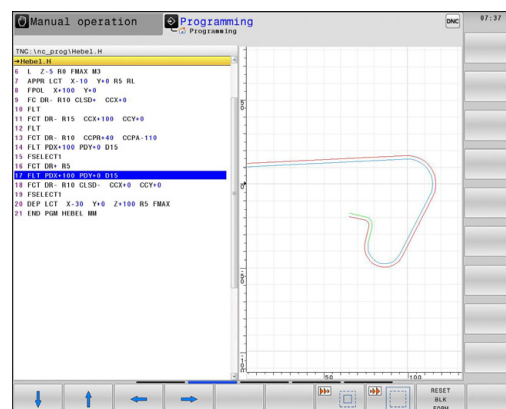
The following functions are available:

Soft key	Function
	Press the desired soft key to move the frame overlay
	
	
	
	Press the soft key to reduce the detail
	Press the soft key to enlarge the detail

The **RESET WORKPIECE BLANK** soft key is used to restore the original section.

You can also use the mouse to change the 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. If you simultaneously press the shift key, you can only move 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 TNC zooms in on the defined area.
- To rapidly magnify or reduce any area: Rotate the mouse wheel backwards or forwards.



4.7 Error messages

4.7 Error messages

Display of errors

The TNC generates error messages when it detects problems such as:

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

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

If a rare "processor check error" should occur, the TNC automatically opens the error window. You cannot remove such an error. Shut down the system and restart the TNC.

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

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

Open the error window



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

Closing the error window



- ▶ Press the **END** soft key—or

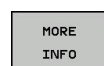


- ▶ Press the **ERR** key. The TNC closes the error window.

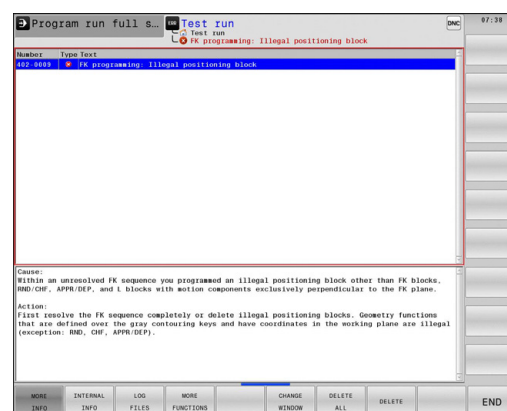
Detailed error messages

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

- Open the error window



- Information on the error cause and corrective action: Position the highlight on the error message and press the **soft key**. The TNC opens a window with information on the error cause and corrective action.
- Leave Info: Press the **soft key** again



INTERNAL INFO soft key

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

- Open the error window.



- Detailed information about the error message: Position the highlight on the error message and press the **INTERNAL INFO** soft key. The TNC opens a window with internal information about the error
- To exit Details, press the **INTERNAL INFO** soft key again.

4.7 Error messages

Clearing errors

Clearing errors outside of the error window

- CE

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

In some operating modes (such as the Editing mode), the CE button cannot be used to clear the error, since the button is reserved for other functions.

Deleting errors

- ▶ Open the error window
- DELETE

▶ Clear individual errors: Position the highlight on the error message and press the **DELETE** soft key.
- DELETE ALL

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

If the cause of the error has not been removed, the error message cannot be deleted. In this case, the error message remains in the window.

Error log

The TNC stores errors and important events (e.g. system startup) in an error log. The capacity of the error log is limited. If the log is full, the TNC uses a second file. If this is also full, the first error log is deleted and written to again, and so on. To view the error history, switch between **CURRENT FILE** and **PREVIOUS FILE**.

- ▶ 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

▶ If you need the previous log file: Press the **PREVIOUS FILE** soft key.
- CURRENT FILE

▶ If you need the current log file: 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 TNC stores keystrokes and important events (e.g. system startup) 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 second file becomes full, the first keystroke log is cleared and written to again, and so on. To view the keystroke history, switch between **CURRENT FILE** and **PREVIOUS FILE**.








- | | |
|---|--|
| <div style="border: 1px solid black; padding: 2px; text-align: center; width: fit-content;">LOG
FILES</div> | ▶ Press the LOG FILES soft key |
| <div style="border: 1px solid black; padding: 2px; text-align: center; width: fit-content;">KEYSTROKE
LOG</div> | ▶ Open the keystroke log file: Press the KEYSTROKE LOG soft key |
| <div style="border: 1px solid black; padding: 2px; text-align: center; width: fit-content;">PREVIOUS
FILE</div> | ▶ If you need the previous log file: Press the PREVIOUS FILE soft key |
| <div style="border: 1px solid black; padding: 2px; text-align: center; width: fit-content;">CURRENT
FILE</div> | ▶ If you need the current log file: Press the CURRENT FILE soft key |

The TNC 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.

4.7

Error messages

Overview of the keys and soft keys for viewing the logs

Soft key/ Keys	Function
	Go to beginning of keystroke log
	Go to end of keystroke log
	Current keystroke log
	Previous keystroke log
	Up/down one line
	
	Return to main menu

Informational texts

After a faulty operation, such as pressing a key without function or entering a value outside of the valid range, the TNC displays a (green) text in the header, informing you that the operation was not correct. The TNC clears this informational text upon the next valid input.

Saving service files

If necessary, you can save the "Current status of the TNC," 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 operation).

If you repeat the "Save service files" function with the same file name, the previously saved group of service data files is overwritten. To avoid this, use another file name when you repeat the function.

Saving service files

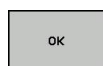
- Open the error window.



- Press the **LOG FILES** soft key.



- Press the **SAVE SERVICE FILES** soft key: The TNC opens a pop-up window in which you can enter a name for the service file.



- Save the service files: Press the **OK** soft key.

Calling the TNCguide help system

You can call the TNC'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.



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



- Call the help for HEIDENHAIN error messages



- Call the help for HEIDENHAIN error messages, if available

Programming: Programming aids

4.8 TNCguide context-sensitive help system

4.8 TNCguide context-sensitive help system

Application



Before you can use the TNCguide, you need to download the help files from the HEIDENHAIN home page (see "Downloading current help files", page 167).

The **TNCguide** context-sensitive help system includes the user documentation in HTML format. The TNCguide is called with the **HELP** key, and the TNC 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 TNC always tries to start the TNCguide in the language that you have selected as the conversational language on your TNC. If the files with this language are not yet available on your TNC, it automatically opens the English version.

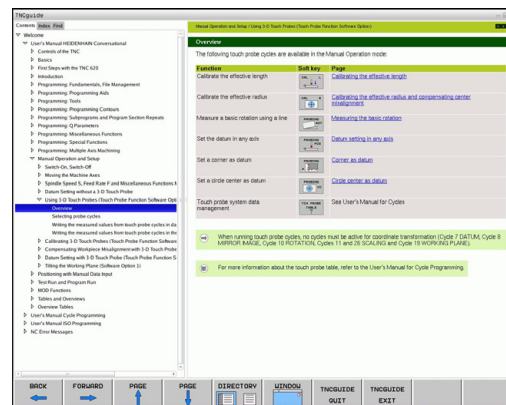
The following user documentation is available in the TNCguide:

- Conversational Programming User's Manual (**BHBKlartext.chm**)
- DIN/ISO User's Manual (**BHBIso.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 the TNCguide

Calling the TNCguide

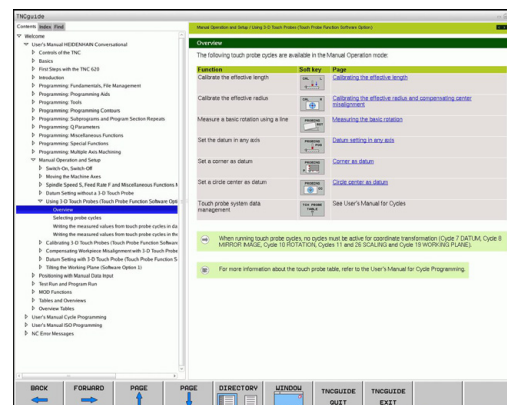
There are several ways to start the TNCguide:

- ▶ Press the **HELP** key if the TNC is not already showing an error message
- ▶ Click the help symbol at the lower right of the screen beforehand, then click the appropriate soft keys
- ▶ Use the file manager to open a help file (.chm file). The TNC can open any .chm file, even if it is not saved on the TNC's internal memory



If one or more error messages are waiting for your attention, the TNC shows the help directly associated with the error messages. To start the **TNCguide**, you first have to acknowledge all error messages.

When the help system is called on the programming station, the TNC starts 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 TNC 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, and click: The TNC opens the TNCguide. If no specific part of the help is assigned to the selected soft key, the TNC opens the book file **main.chm**, in which you can use the search function or the navigation to find the desired explanation manually

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

Programming: Programming aids













4.8 TNCguide context-sensitive help system





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 the TNCguide through keys and soft keys. The following table contains an overview of the corresponding key functions.

Soft key	Function
	<ul style="list-style-type: none"> ■ If the table of contents at left is active: Select the entry above it or below it
	<ul style="list-style-type: none"> ■ If the text window at right is active: Move the page downward or upward if texts or graphics are not shown completely
	<ul style="list-style-type: none"> ■ 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
	<ul style="list-style-type: none"> ■ If the table of contents at left is active: Close the table of contents ■ If the text window at right is active: No function
	<ul style="list-style-type: none"> ■ 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
	<ul style="list-style-type: none"> ■ 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
	<ul style="list-style-type: none"> ■ If the table of contents at left is active: Select the entry above it or below it
	<ul style="list-style-type: none"> ■ If the text window at right is active: Jump to next link
	Select the page last shown
	Page forward if you have used the "select page last shown" function
	Move up by one page
	Move down by one page

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

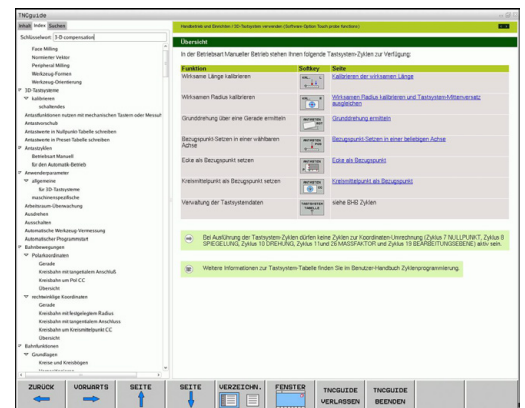
Subject index

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
- ▶ Activate the **Keyword** input field
- ▶ Enter the word for the desired subject and the TNC synchronizes the index and creates a list in which you can find the subject more easily, or
- ▶ Use the arrow key to highlight the desired keyword
- ▶ Use the **ENT** key to call the information on the selected keyword



Programming: Programming aids

4.8 TNCguide context-sensitive help system

Full-text search

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

The left side is active.



- ▶ Select the **Find** tab
- ▶ Activate the **Find:** input field
- ▶ Enter the desired word and confirm with the **ENT** key: The TNC lists all sources containing the word
- ▶ Use the arrow key to highlight 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 (by mouse or by selecting it and then pressing the space key), the TNC searches only through headings and ignores the body text.

Downloading current help files

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

- ▶ Documentation and information
- ▶ Documentation
- ▶ User Documentation
- ▶ TNCguide
- ▶ Select the desired language
- ▶ TNC Controls
- ▶ Series, e.g. TNC 600
- ▶ Desired NC software number, e.g. TNC 640 (34059x-04)
- ▶ Select the desired language version from the **TNCguide online help** table
- ▶ Download the ZIP file and unpack it
- ▶ Move the unzipped CHM files to the TNC in the **TNC:\tncguide\en** directory or into the respective language subdirectory (see also the following table)



If you want to use TNCremo to transfer the .chm files to the TNC, then in the **Extras > Configuration > Mode > Transfer in binary format** menu item you have to enter the extension **.CHM**.

Programming: Programming aids

4.8 TNCguide context-sensitive help system

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

5

**Programming:
Tools**

5

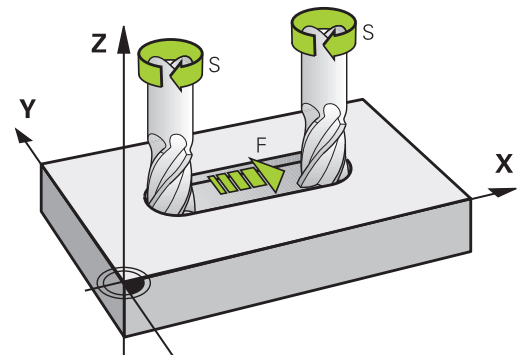
Programming: Tools

5.1 Entering tool-related data

5.1 Entering tool-related data

Feed rate **F**

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



Input

You can enter the feed rate in the **TOOL CALL** block (tool call) and in every positioning block (see "Creating the program blocks with the path function keys", page 210). In millimeter-programs you enter the feed rate **F** in mm/min, and in inch-programs, for reasons of resolution, in 1/10 inch/min. Alternatively, with the corresponding soft keys, you can also define the feed rate in mm per revolution (mm/rev) **FU** or in mm per tooth (mm/tooth) **FZ**.

Rapid traverse

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



To move your machine at rapid traverse, you can also program the corresponding numerical value, e.g. **F30000**. Unlike **F MAX**, 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 a block with a different feed rate is reached. **F MAX** is only effective in the block in which it is programmed. After the block with **F MAX** is executed, the feed rate will return to the last feed rate entered as a numerical value.

Changing during program run

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

Spindle speed S

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

Programmed change

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



- ▶ Program a tool call: Press the **TOOL CALL** key
- ▶ Ignore the dialog question for **Tool number ?** with the **NO ENT** key
- ▶ Ignore the dialog question for **Working spindle axis X/Y/Z ?** with the **NO ENT** key
- ▶ Enter the new spindle speed for the dialog question **Spindle speed S= ?**, and confirm with **END**, or switch via the **VC** soft key to entry of the cutting speed.

Changing during program run

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

Programming: Tools

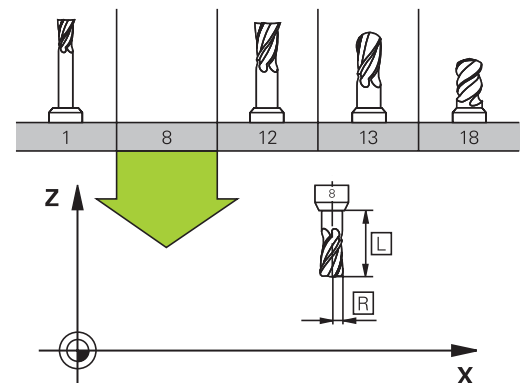
5.2 Tool data

5.2 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 TNC 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.

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



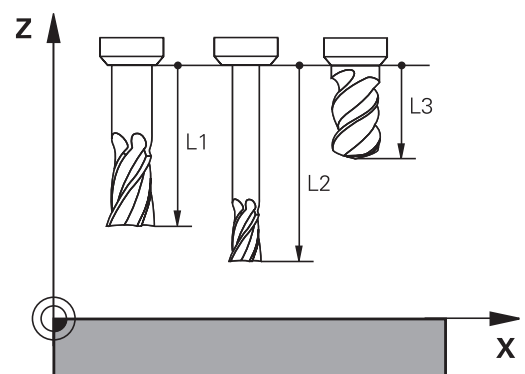
Permitted special characters: # \$ % & , - . 0 1 2 3 4 5
6 7 8 9 @ A B C D E F G H I J K L M N O P Q R S T U
V W X Y Z _

Impermissible characters: <blank space> " ' () * + : ;
< = > ? [/] ^ ` 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 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 TNC 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**, **DR2**>0). If you are programming the machining data with an allowance, enter the oversize value in the **TOOL CALL** block of the part program.

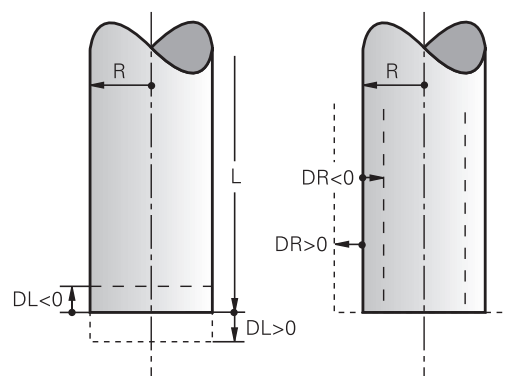
A negative delta value describes a tool undersize (**DL**, **DR**, **DR2**<0). An undersize is entered in the tool table for wear.

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

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



Delta values from the tool table influence the graphical representation of the clearing simulation. Delta values from the **TOOL CALL** block influence the position display depending on the machine parameter **progToolCallIDL**.



Entering tool data into the program



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

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

- Select the tool definition: Press the **TOOL DEF** key



- **Tool number**: Each tool is uniquely identified by its tool number
- **Tool length**: Compensation value for the tool length
- **Tool radius**: Compensation value for the tool radius



In the programming dialog, you can transfer the value for tool length and tool radius directly into the input line by pressing the desired axis soft key.

Example

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


Enter tool data into the table

You can define and store up to 32767 tools and their tool data in a tool table. Also see the editing functions later in this Chapter. In order to be able to assign various compensation data to a tool (indexing tool number), insert a line and extend the tool number by a dot and a number from 1 to 9 (e.g. **T 5.2**).

You must use tool tables if

- you wish to use indexed tools such as stepped drills with more than one length compensation value
- your machine tool has an automatic tool changer
- you want to fine-rough the contour with Cycle 22, (see "User's Manual for Cycle Programming, ROUGH-OUT")
- you want to work with Cycles 251 to 254 (see "User's Manual for Cycle Programming," Cycles 251 to 254)



if you create or manage further tool tables, the file name has to start with a letter.

You can select either list view or form view for tables via the "Screen layout" key.

when you open the tool table you can also change its layout

Tool table: Standard tool data

Abbr.	Inputs	Dialog
T	Number by which the tool is called in the program (e.g. 5, indexed: 5.2)	-
NAME	Name by which the tool is called in the program (no more than 32 characters, all capitals, no spaces)	Tool name?
L	Compensation value for tool length L	Tool length?
R	Compensation value for the tool radius R	Tool radius?
R2	Tool radius R2 for toroid cutters (only for 3-D radius compensation or graphical representation of a machining operation with spherical cutters)	Tool radius 2?
DL	Delta value for tool length L	Tool length oversize?
DR	Delta value for tool radius R	Tool radius oversize?
DR2	Delta value for tool radius R2	Tool radius oversize 2?
ANGLE	Maximum plunge angle of the tool for reciprocating plunge-cut in Cycles 22 and 208	Maximum plunge angle?
TL	Set tool lock (TL : for Tool Locked)	Tool locked? Yes=ENT/No=NO ENT
RT	Number of a replacement tool – if available – as replacement tool (RT : for R eplacement T ool; also see TIME2) An empty field or input 0 means no replacement tool has been defined.	Replacement tool?
TIME1	Maximum tool life in minutes. This function can vary depending on the individual machine tool. Your machine manual provides more information	Maximum tool age?
TIME2	Maximum tool life in minutes during TOOL CALL : If the current tool life reaches or exceeds this value, the TNC changes the tool during the next TOOL CALL (see also CUR_TIME).	Max. tool age for TOOL CALL?
CUR_TIME	Current age of the tool in minutes: The TNC automatically counts the current tool life (CUR_TIME : for CUR rent TIME). A starting value can be entered for used tools	Current tool age?

Programming: Tools

5.2 Tool data

Abbr.	Inputs	Dialog
TYPE	Tool type: Press the ENT key to edit the field; the GOTO key opens a window in which you can select the tool type. You can assign tool types to specify the display filter settings such that only the selected type is visible in the table	Tool type?
DOC	Comment on tool (up to 32 characters)	Tool comment?
PLC	Information on this tool that is to be sent to the PLC	PLC status?
LCUTS	Tooth length of the tool for Cycle 22	Tooth length in the tool axis?
PTYP	Tool type for evaluation in the pocket table Function is defined by the machine tool builder. The machine tool documentation provides further information	Tool type for pocket table?
NMAX	Limit the spindle speed for this tool. The programmed value is monitored (error message) as well as an increase in the shaft speed via the potentiometer. Function inactive: Enter -. Input range: 0 to +999999, if function not active: enter -	Maximum shaft speed [rpm]
LIFTOFF	Definition of whether the TNC should retract the tool in the direction of the positive tool axis at an NC stop in order to avoid leaving dwell marks on the contour. If Y is defined, the TNC retracts the tool from the contour, provided that this function was activated in the NC program with M148 see "Automatically retract tool from the contour at an NC stop: M148", page 386.	Retraction permissible? Yes=ENT/No=NOENT
TP_NO	Reference to the number of the touch probe in the touch-probe table	Number of the touch probe
T-ANGLE	Point angle of the tool. Is used by the Centering cycle (Cycle 240) in order to calculate the centering depth from the diameter entry	point angle
PITCH	Thread pitch of the tool. Used by tapping cycles (Cycle 206, Cycle 207 and Cycle 209). A positive algebraic sign means a right-hand thread.	Tool thread pitch?
AFC	Control setting for the adaptive feed control AFC that you have defined in the NAME column of the AFC.TAB table. Apply the feedback-control strategy with the ASSIGN AFC CONTROL SETTING soft key (3rd soft-key row) Input range: 10 characters max.	Feedback-control strategy
LAST_USE	Date and time that the tool was last inserted via TOOL CALL	Date/time of last tool call
ACC	Activate or deactivate active chatter control for the respective tool (page 412). Input range: N (inactive) and Y (active)	ACC active? Yes=ENT/No=NOENT

Tool table: Tool data required for automatic tool measurement

For a description of the cycles for automatic tool measurement, see the User's Manual for Cycle Programming.

Abbr.	Inputs	Dialog
CUT	Number of teeth (99 teeth maximum)	Number of teeth?
LTOL	Permissible deviation from tool length L for wear detection. If the entered value is exceeded, the TNC locks the tool (status L). Input range: 0 to 0.9999 mm	Wear tolerance: length?
RTOL	Permissible deviation from tool radius R for wear detection. If the entered value is exceeded, the TNC locks the tool (status L). Input range: 0 to 0.9999 mm	Wear tolerance: radius?
R2TOL	Permissible deviation from tool radius R2 for wear detection. If the entered value is exceeded, the TNC locks the tool (status L). Input range: 0 to 0.9999 mm	Wear tolerance: Radius 2?
DIRECT	Cutting direction of the tool for measuring the tool during rotation	Cutting direction? M4=ENT/M3=NOENT
R-OFFS	Tool radius measurement: Tool offset between stylus center and tool center. Default setting: No value entered (offset = tool radius)	Tool offset: radius?
L-OFFS	Tool length measurement: Tool offset in addition to offsetToolAxis between upper surface of stylus and lower surface of tool. Default: 0	Tool offset: length?
LBREAK	Permissible deviation from tool length L for breakage detection. If the entered value is exceeded, the TNC locks the tool (status L). Input range: 0 to 3.2767 mm	Breakage tolerance: length?
RBREAK	Permissible deviation from tool radius R for breakage detection. If the entered value is exceeded, the TNC locks the tool (status L). Input range: 0 to 0.9999 mm	Breakage tolerance: radius?

Editing the tool table

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

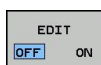
Other tool tables that are to be archived or used for test runs are given different file names with the extension .T. By default, for the **Test Run** and **Programming** modes the TNC also uses the TOOL.T tool table. In the **Test Run** mode, press the **TOOL TABLE** soft key to edit it.

To open the tool table TOOL.T:

- ▶ Select any machine operating mode



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



- ▶ Set the **EDIT** soft key to **ON**

NAME	L	R	R2	DL	DR
0/MALWERKZEUG	0	0	0	0	0
1.02	30	1	0	0	0
2.04	40	2	0	0	0
3.06	50	3	0	0	0
4.08	60	4	0	0	0
5.010	60	5	0	0	0
6.012	60	6	0	0	0
7.014	70	7	0	0	0
8.016	80	8	0	0	0
9.018	90	9	0	0	0
10.020	90	10	0	0	0
11.022	90	11	0	0	0
12.024	90	12	0	0	0
13.026	90	13	0	0	0
14.028	100	14	0	0	0
15.030	100	15	0	0	0
16.032	100	16	0	0	0
17.034	100	17	0	0	0
18.036	100	18	0	0	0
19.038	100	19	0	0	0
20.040	100	20	0	0	0
21.042	100	5	5	0	0
22.044	120	22	0	0	0
23.046	120	23	0	0	0
24.048	120	24	0	0	0
25.050	120	25	0	0	0
26.052	120	26	0	0	0

Displaying only specific tool types (filter setting)

- ▶ Press the **TABLE FILTER** soft key (fourth soft-key row)
- ▶ Select the tool type by pressing a soft key: The TNC only shows tools of the type selected
- ▶ Cancel the filter: Press the **SHOW ALL** soft key



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

Hiding or sorting the tool table columns

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

- ▶ Press the **SORT/HIDE COLUMNS** soft key (fourth soft-key row)
- ▶ Select the appropriate column name with the arrow key
- ▶ Press the **HIDE COLUMN** soft key to remove this column from the table layout

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

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

You can use a connected mouse or the TNC keyboard to navigate in the form. Navigation using the TNC keyboard:



- ▶ Press the navigation keys to go to the input fields. Use the arrow keys to navigate within an input field. To open pop-down menus, press the **GOTO** key.



With the **Fix number of columns** function, you can define how many columns (0 -3) are fixed to the left screen edge. These columns are also displayed if you navigate in the table to the right.

Programming: Tools

5.2 Tool data






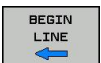
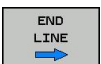
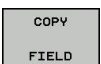
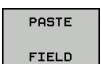
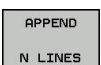
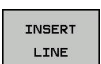

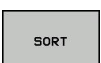
Opening any other tool table





- Select the **Programming** mode of operation



- Call the File Manager
- Select a file or enter a new file name. Conclude your entry with the **ENT** key or the **SELECT** soft key

When you have opened the tool table, you can edit the tool data by moving the cursor to the desired position in the table with the arrow keys or the soft keys. You can overwrite the stored values, or enter new values at any position. Additional functions are illustrated in the table below.

Soft key	Editing functions for tool tables
	Select beginning of table
	Select end of table
	Select previous page in table
	Select next page in table
	Find the text or number
	Move to beginning of line
	Move to end of line
	Copy highlighted field
	Insert copied field
	Add the entered number of lines (tools) at the end of the table
	Adding a row with tool number for entering
	Delete current line (tool)
	Sort the tools according to the content of a column

Soft key	Editing functions for tool tables
	Show all drills in the tool table
	Show all cutters in the tool table
	Show all taps/thread cutters in the tool table
	Show all touch probes in the tool table

Exiting any other tool table

- Call the file manager and select a file of a different type, such as a part program

Tool table for turning tools

With the management of turning tools, other geometric descriptions are considered than with milling or drilling tools. To be able to execute tool radius compensation, for example, you have to define the tool radius. To support these definitions, the TNC provides a special tool management for turning tools, see "Tool data", page 499.

Importing tool tables



The machine manufacturer can adapt the **IMPORT TABLE** function. Refer to your machine manual.

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

- ▶ Save the tool table of the iTNC 530 to the **TNC:\table** directory
- ▶ Select the **Programming** mode of operation. **Programming**
- ▶ Call the file manager: Press the **PGM MGT** key
- ▶ Move the highlight to the tool table you want to import
- ▶ Press the **MORE FUNCTIONS** soft key
- ▶ Shift the soft-key row
- ▶ Select the **IMPORT TABLE** soft key: The TNC inquires whether you really want to overwrite the selected tool table
- ▶ Do not overwrite the file: Press the **CANCEL** soft key, or
- ▶ Overwrite the file: Press the **OK** soft key
- ▶ Open the converted table and check its contents



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

The TNC changes a comma in the tool name to a period during import.

The TNC overwrites the selected tool table when running the **IMPORT TABLE** function. To avoid losing data, be sure to make a backup copy of your original tool table before importing it!

The procedure for copying tool tables using the TNC file manager is described in the section on file management (see "Copying a table", page 123).

When tool tables are imported from an iTNC 530, all existing tools are imported along with their corresponding tool type. Nonexistent tool types are imported as type 0 (MILL). Check the tool table after the import.

Pocket table for tool changer



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

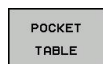
For automatic tool changing you need the a pocket table. You manage the assignment of your tool changer in the pocket table. The pocket table is in the **TNC:\TABLE** directory. The machine tool builder can adapt the name, path and content of the pocket table. You can also select various layouts using soft keys in the **TABLE FILTER** menu.

T	NAME	L	R	R2	DL	DR
0	SMALL WERKZEUGE	0	0	0	0	0
100		20	11	0	0	0
204		40	2	0	0	0
306		50	3	0	0	0
408		50	4	0	0	0
5010		60	5	0	0	0
6012		60	6	0	0	0
7014		70	7	0	0	0
8016		80	8	0	0	0
9018		90	9	0	0	0
10020		90	10	0	0	0
11022		90	11	0	0	0
12024		90	12	0	0	0
13026		90	13	0	0	0
14028		100	14	0	0	0
15030		100	15	0	0	0
16032		100	16	0	0	0
17034		100	17	0	0	0
18036		100	18	0	0	0
19038		100	19	0	0	0
20040		100	20	0	0	0
21042		100	5	5	0	0
22044		120	22	0	0	0
23046		120	23	0	0	0
24048		120	24	0	0	0
25050		120	25	0	0	0
26052		120	26	0	0	0

Editing a pocket table in a Program Run operating mode



- To select the tool table, press the **TOOL TABLE** soft key.



- Select the pocket table: Press the **POCKET TABLE** soft key




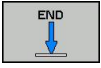




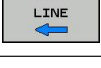
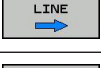


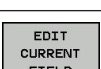

- Set the **EDIT** soft key to **ON**. On your machine this might not be necessary or even possible. Refer to your machine manual.

Selecting a pocket table in the Programming mode of operation



- ▶ Call the File Manager
- ▶ Display the file types: Press the **SHOW ALL** soft key
- ▶ Select a file or enter a new file name. Conclude your entry with the **ENT** key or the **SELECT** soft key

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

Soft key	Editing functions for pocket tables
	Select beginning of table
	Select end of table
	Select previous page in table
	Select next page in table
	Reset pocket table
	Reset tool number column T
	Go to beginning of the line
	Go to end of the line
	Simulate a tool change
	Select a tool from the tool table: The TNC shows the contents of the tool table. Use the arrow keys to select a tool, press OK to transfer it to the pocket table
	Edit the current field
	Sort the view



The machine manufacturer defines the features, properties and designations of the various display filters. Refer to your machine manual.

Call tool data

A **TOOL CALL** in the part program is defined with the following data:

- ▶ Select the tool call function with the **TOOL CALL** key



- ▶ **Tool number**: Enter the number or name of the tool. The tool must already be defined in a **TOOL DEF** block or in the tool table. With the **TOOL NAME** soft key you can enter a name. With the **QS** soft key you enter a string parameter. The TNC 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**. If you wish to call a tool with other compensation values, also enter the index you defined in the tool table after the decimal point. There is a **SELECT** soft key for calling a window from which you can select a tool defined in the tool table **TOOL.T** directly without having to enter the number or name
- ▶ **Working spindle axis X/Y/Z**: Enter the tool axis
- ▶ **Spindle speed S**: Enter the spindle speed **S** in revolutions per minute (rpm). Instead, you can define the cutting speed **Vc** in meters per minute (m/min). Press the **VC** soft key
- ▶ **Feed rate F**: Enter feed rate **F** in millimeters per minute (mm/min). Alternatively, with the corresponding soft keys, you can also define the feed rate in mm per revolution (mm/rev) **FU** or in mm per tooth (mm/tooth) **FZ**. The feed rate is effective until you program a new feed rate in a positioning or **TOOL CALL** block
- ▶ **Tool length oversize DL**: Enter the delta value for the tool length
- ▶ **Tool radius oversize DR**: Enter the delta value for the tool radius
- ▶ **Tool radius oversize DR2**: Enter the delta value for the tool radius 2



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

You can also search for a tool in the pop-up window. To do so, press the **GOTO** or **SEARCH** soft key and enter the tool number or tool name. With the **OK** soft key you can load the tool into the dialog box.

Example: 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.

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

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

Preselection of tools



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

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

Programming: Tools

5.2 Tool data

Tool change

Automatic tool change



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

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

Automatic tool change if the tool life expires: M101



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

When the specified tool life has expired, the TNC 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 TNC 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.

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

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


Caution: Danger to the workpiece and tool!

Switch off the automatic tool change with **M102** if you are working with special tools (e.g. side mill cutter) because the TNC at first always moves the tool away from the workpiece in tool axis direction.

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 element **BT** (block tolerance)

If you enter the **M101** function, the TNC continues the dialog by requesting the **BT**. Here you define the number of NC blocks (1 - 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 TNC uses the value 1 or, if applicable, a default value defined by the machine manufacturer.



The more you increase the value of **BT**, the smaller will be the effect of an extended program duration through **M101**. Please note that this will delay the automatic tool change!

To calculate a suitable output value for **BT** use the formula **BT = 10 : Average machining time of an NC block in seconds**. Round up to the next odd integer. 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 NC blocks with surface-normal vectors and 3-D compensation

The active radius (**R + DR**) of the replacement tool must not differ from the radius of the original tool. You can enter the delta values (**DR**) either in the tool table or in the **TOOL CALL** block. If there are any deviations, the TNC displays an error message and does not replace the tool. You can suppress this message with the M function **M107**, and reactivate it with **M108**. See also: "Three-dimensional tool compensation (Option #9)", page 474.

Tool usage test



The tool usage test function must be enabled by your machine manufacturer. Refer to your machine manual.



The tool usage test is not available for turning tools.

In order to be able to conduct a tool usage test, tool usage files have to be generated, see page 618

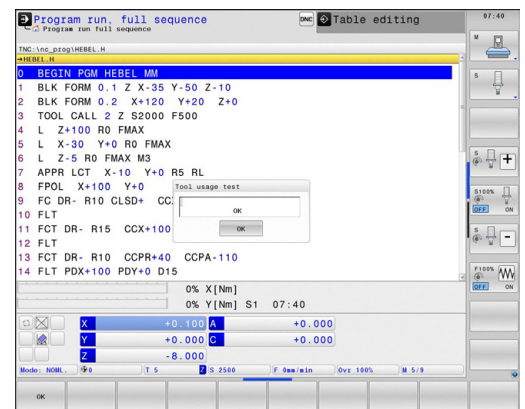
The NC program has to be completely simulated in the **Test Run** operating mode or executed in the **Program Run, Full Sequence** or **Single Block** operating mode.

Applying the tool usage test

Before starting a program in the Program Run mode of operation, you can use the **TOOL USAGE** and **TOOL USAGE TEST** soft keys to check whether the tools being used in the selected program are available and have sufficient remaining service life. The TNC then compares the actual service-life values in the tool table with the nominal values from the tool usage file.

After you have pressed the **TOOL USAGE TEST** soft key, the TNC displays the result of the tool usage test in a pop-up window. To close the pop-up window, press the ENT key.

The TNC saves the tool usage times in a separate file with the extension **pgmname.H.T.DEP**. This file is not visible unless the machine parameter **CfgPgmMgt/dependentFiles** is set to **MANUAL**. The generated tool usage file contains the following information:



Column	Meaning
TOKEN	<ul style="list-style-type: none"> ■ TOOL: Tool usage time per TOOL CALL. The entries are listed in chronological order. ■ TTOTAL: Total usage time of a tool ■ STOTAL: Call of a subprogram; the entries are listed in chronological order ■ TIMETOTAL: The total machining time of the NC program is entered in the WTIME column. In the PATH column the TNC saves the path name of the corresponding NC programs. The TIME column shows the sum of all TIME entries (feed time without rapid traverse movements). The TNC sets all other columns to 0 ■ TOOLFILE: In the PATH column, the TNC saves the path name of the tool table with which you conducted the Test Run. This enables the TNC during the actual tool usage test to detect whether you performed the test run with the TOOL.T

Column	Meaning
TNR	Tool number (-1: No tool inserted yet)
IDX	Tool index
NAME	Tool name from the tool table
TIME	Tool-usage time in seconds (feed time without rapid traverse movements)
WTIME	Tool-usage time in seconds (total usage time between tool changes)
RAD	Tool radius R + Oversize of tool radius DR from the tool table. (in mm)
BLOCK	Block number in which the TOOL CALL block was programmed
PATH	<ul style="list-style-type: none"> ■ TOKEN = TOOL: Path name of the active main program or subprogram ■ TOKEN = STOTAL: Path name of the subprogram
T	Tool number with tool index
OVRMAX	Maximum feed rate override that occurred during machining. During Test Run the TNC enters the value 100 (%)
OVRMIN	Minimum feed rate override that occurred during machining. During Test Run the TNC enters the value -1
NAMEPROG	<ul style="list-style-type: none"> ■ 0: The tool number is programmed ■ 1: The tool name is programmed

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

- The highlight in the pallet file is on a pallet entry: The TNC runs the tool usage test for the entire pallet
- The highlight in the pallet file is on a pallet entry: The TNC runs the tool usage test for the entire pallet

Tool management (Option #93)



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























With the tool management, your machine tool builder can provide many functions with regard to tool handling. Examples:

- Easily readable and, if you desired, adaptable representation of the tool data in fillable forms
- Any description of the individual tool data in the new table view
- Mixed representation of data from the tool table and the pocket table
- Fast sorting of all tool data by mouse
- Use of graphic aids, e.g. color coding of tool or magazine status
- Program-specific list of all available tools
- Program-specific usage sequence of all tools
- Copying and pasting of all tool data pertaining to a tool
- Graphic depiction of tool type in the table view and in the detail view for a better overview of the available tool types

The screenshot shows a software interface titled 'Expanded tool management' with a 'Programming' tab. It displays a table with columns: 'Tool', 'Name', 'Type', 'Magazine', 'Tool life', and 'Remaining life'. The table lists various tools (e.g., D2, D4, D8, D10, D12, D14, D16, D18, D20, D22, D24, D26, D28, D30, D32, D34, D36, D38, D40, D42, D44, D46, D48, D50, D52, D54, D56, D58, D60, D62) and their corresponding magazine numbers and tool life values. The interface includes navigation buttons at the bottom: 'BEGIN', 'END', 'PAGE', 'MAGAZINE MANAGEMENT', 'FORM FOR TOOL', and 'END'.

Tool	Name	Type	Magazine	Tool life	Remaining life
1	D2	0	0	Not monitored	0
2	D4	0	0	Not monitored	0
3	D8	0	0	Not monitored	0
4	D10	0	0	Not monitored	0
5	D12	0	0	Not monitored	0
6	D14	0	0	Not monitored	0
7	D16	0	0	Not monitored	0
8	D18	0	0	Not monitored	0
9	D20	0	0	Not monitored	0
10	D22	0	0	Not monitored	0
11	D24	0	0	Not monitored	0
12	D26	0	0	Not monitored	0
13	D28	0	0	Not monitored	0
14	D30	0	0	Not monitored	0
15	D32	0	0	Not monitored	0
16	D34	0	0	Not monitored	0
17	D36	0	0	Not monitored	0
18	D38	0	0	Not monitored	0
19	D40	0	0	Not monitored	0
20	D42	0	0	Not monitored	0
21	D44	0	0	Not monitored	0
22	D46	0	0	Not monitored	0
23	D48	0	0	Not monitored	0
24	D50	0	0	Not monitored	0
25	D52	0	0	Not monitored	0
26	D54	0	0	Not monitored	0
27	D56	0	0	Not monitored	0
28	D58	0	0	Not monitored	0
29	D60	0	0	Not monitored	0
30	D62	0	0	Not monitored	0

Available tool types

Icon	Tool type
	Undefined, ****
	Milling cutter, MILL
	Drill, DRILL
	Tap, TAP
	Center drill, CENT
	Turning Tool, TURN
	Touch probe, TCHP
	Ream, REAM
	Countersink, CSINK
	Piloted counterbore(TSINK), TSINK
	Boring tool, BOR
	Back boring tool, BCKBOR
	Thread mill, GF
	Thread mill w/ countersink, GSF
	Thread mill w/ single thread, EP
	Thread mill w/ indxbl insert, WSP
	Thread milling drill, BGF
	Circular thread mill, ZBGF
	Roughing cutter (MILL_R), MILL_R
	Finishing cutter (MILL_F), MILL_F
	Rough/finish cutter, MILL_RF
	Floor finisher(MILL_FD), MILL_FD

Icon

Tool type



Side finisher (MILL_FS),MILL_FS



Face milling cutter,MILL_FACE

Calling the File Manager



The tool management call can differ as described below. Refer to your machine manual.



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



- ▶ Scroll through the soft-key row



- ▶ Select the **TOOL MANAGEMENT** soft key: The TNC goes into the new table view (see figure at right)

Tools	POCKET	TOOLING	TOOL	USAGE	SEQUENCE	TOOL LIFE	REMAINING LIFE
1	002	0	0	0	0	Not monitored	0
2	004	0	0	0	0	Not monitored	0
3	006	0	0	0	0	Not monitored	0
4	008	0	0	0	0	Not monitored	0
5	010	0	0	0	0	Not monitored	0
6	012	0	0	0	0	Not monitored	0
7	014	0	0	0	0	Not monitored	0
8	016	0	0	0	0	Not monitored	0
9	018	0	0	0	0	Not monitored	0
10	020	0	0	0	0	Not monitored	0
11	022	0	0	0	0	Not monitored	0
12	024	0	0	0	0	Not monitored	0
13	026	0	0	0	0	Not monitored	0
14	028	0	0	0	0	Not monitored	0
15	030	0	0	0	0	Not monitored	0
16	032	0	0	0	0	Not monitored	0
17	034	0	0	0	0	Not monitored	0
18	036	0	0	0	0	Not monitored	0
19	038	0	0	0	0	Not monitored	0
20	040	0	0	0	0	Not monitored	0
21	042	0	0	0	0	Not monitored	0
22	044	0	0	0	0	Not monitored	0
23	046	0	0	0	0	Not monitored	0
24	048	0	0	0	0	Not monitored	0
25	050	0	0	0	0	Not monitored	0
26	052	0	0	0	0	Not monitored	0
27	054	0	0	0	0	Not monitored	0
28	056	0	0	0	0	Not monitored	0
29	058	0	0	0	0	Not monitored	0
30	060	0	0	0	0	Not monitored	0
31	062	0	0	0	0	Not monitored	0
32	064	0	0	0	0	Not monitored	0
33	066	0	0	0	0	Not monitored	0
34	068	0	0	0	0	Not monitored	0
35	070	0	0	0	0	Not monitored	0

In the new view, the TNC presents all tool information in the following four tabs:

- **Tools:** Tool specific information
- **Pockets:** Pocket-specific information
- **Assembly list:** List of all tools in the NC program that is selected in the Program Run mode (only if you have already created a tool usage file, see "Tool usage test", page 190)
- **T usage sequence:** List of the sequence of all tools that are inserted in the program selected in the Program Run mode (only if you have already made a tool usage file, see "Tool usage test", page 190)



You can edit the tool data only in the form view, which you can activate by pressing the **FORM FOR TOOL** soft key or the **ENT** key for the currently highlighted tool.





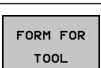

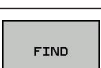




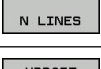


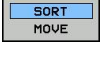
If you use tool management without a mouse, then you can activate and deactivate functions with the "-/+" check box.

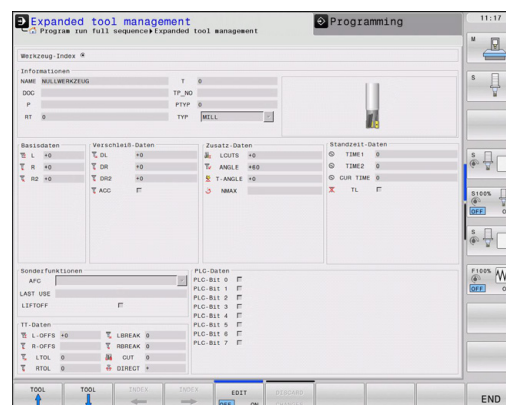
In the tool management, use the **GOTO** soft key to search for the tool number or pocket number.

Operating the Tool Manager

The tool management can be operated by mouse or with the keys and soft keys:

Soft key Editing functions of tool management

	Select the table start
	Select the table end
	Select the previous page in the table
	Select the next page in the table
	Call the form view of the marked tool. Alternative function: Press the ENT key
	Go to the next tab: Tools, Pockets, Tooling list, T usage order
	Search function (Find): Here you can select the column to be searched and the search term via a list or by entering it
	Import tools
	Export tools
	Delete marked tools
	Add several lines at end of table
	Update table view
	Show programmed-tools column (if Pockets tab is active)
	Define the settings: <ul style="list-style-type: none"> ■ SORT COLUMN active: Click the column header to sort the content of the column ■ MOVE COLUMN active: The column can be moved by drag and drop
	Reset the manually changed settings (moved columns) to original condition












Programming: Tools

5.2 Tool data

In addition, you can perform the following functions by mouse:

- Sorting function. You can sort the data in ascending or descending order (depending on the active setting) by clicking a column of the table head.
- Arrange columns. You can arrange the columns in any sequence you want by clicking a column of the table head and then moving it with the mouse key pressed down. The TNC does not save the current column sequence when you exit the tool management (depending on the active setting)
- Show miscellaneous information in the form view. The TNC displays tool tips when you leave the mouse pointer on an active input field for more than a second and when you have set the **EDIT ON/OFF** soft key to **ON**.

If the form view is active, the following functions are available to you:

Soft key	Editing functions form view
	Select the tool data of the previous tool
	Select the tool data of the next tool
	Select previous tool index (only active if indexing is enabled)
	Select the next tool index (only active if indexing is enabled)
	Discard all changes made since the form was last called ("Undo" function)
	Insert a line (tool index) (2nd soft-key row)
	Delete a line (tool index) (2nd soft-key row)
	Copy the tool data of the selected tool (2nd soft-key row)
	Insert the copied tool data in the selected tool (2nd soft-key row)

Importing tool data

Using this function you can simply import tool data that you have measured externally on a presetting device, for example. The file to be imported must have the CSV format (Comma Separated Values). The **CSV** file format describes the structure of a text file for exchanging simply structured data. Accordingly, the import file must have the following structure:

- **Row 1:** In the first line you define the column names in which the data defined in the subsequent lines is to be placed. The column names are separated from each other by commas.
- **Other lines:** All the other lines contain the data that you wish to import into the tool table. The order of the data must match the order of the column names in Line 1. The data is separated by commas, decimal numbers are to be defined with a decimal point.

Follow the steps outlined below for importing:

- ▶ Copy the tool table to be imported to the hard disk of the TNC in the **TNC:\systems\tooltab** directory
- ▶ Start expanded tool management
- ▶ Select the **IMPORT TOOL** soft key in the Tool Management: The TNC shows a pop-up window with the CSV files stored in the **TNC:\systems\tooltab** directory
- ▶ Use the arrow keys or mouse to select the file to be imported and confirm with the **ENT** key: The TNC shows the content of the CSV file in a pop-up window
- ▶ Start import procedure with **START** soft key



- The CSV file to be imported must be stored in the **TNC:\system\tooltab** directory.
- If you import the tool data of tools whose numbers are in the pocket table, the TNC issues an error message. You can then decide whether you want to skip this data record or insert a new tool. The TNC inserts a new tool into the first empty line of the tool table.
- Make sure that the column designations are specified correctly, see "Enter tool data into the table", page 174.
- You can import any tool data, the associated data record does not have to contain all the columns (or data) of the tool table.
- The column names can be in any order, the data must be defined in the corresponding order.

Programming: Tools

5.2 Tool data

Sample import file:

T,L,R,DL,DR	Line 1 with column names
4,125.995,7.995,0,0	Line 2 with tool data
9,25.06,12.01,0,0	Line 3 with tool data
28,196.981,35,0,0	Line 4 with tool data

Exporting tool data

Using this function you can simply export tool data to read it into the tool database of your CAM system, for example. The TNC stores the exported file in the CSV format (Comma Separated Values). The **CSV** file format describes the structure of a text file for exchanging simply structured data. The export file has the following structure:

- **Line 1:** In the first line the TNC stores the column names of all the relevant tool data to be defined. The column names are separated from each other by commas.
- **Further lines:** All the other lines contain the data of the tools that you have exported. The order of the data matches the order of the column names in Line 1. The data is separated by commas, the TNC outputs decimal numbers with a decimal point.

Follow the steps outlined below for exporting:

- ▶ In the tool management you use the arrow keys or mouse to mark the tool data that you wish to export
- ▶ Select the **EXPORT TOOL** soft key, the TNC shows a pop-up window: specify the name for the CSV file, confirm with the **ENT** key
- ▶ Press the **START** soft key to start the export process: The TNC shows the status of the delete export process in a pop-up window
- ▶ Terminate the export process by pressing the **END** key or soft key



The TNC always stores the exported CSV file in the **TNC:\system\tooltab** directory.

Deleting marked tool data

Using this function you can simply delete tool data that you no longer need.

Follow the steps outlined below for deleting:

- ▶ In the tool management you use the arrow keys or mouse to mark the tool data that you wish to delete
- ▶ Select the **DELETE MARKED TOOLS** soft key and the TNC shows a pop-up window listing the tool data to be deleted
- ▶ Press the **START** soft key to start the delete process: The TNC shows the status of the delete process in a pop-up window
- ▶ Terminate the delete process by pressing the **END** key or soft key



- The TNC deletes all the data of all the tools selected. Make sure that you really no longer need the tool data, because there is no Undo function available.
- You cannot delete the tool data of tools still stored in the pocket table. First remove the tool from the magazine.

Programming: Tools

5.3 Tool compensation

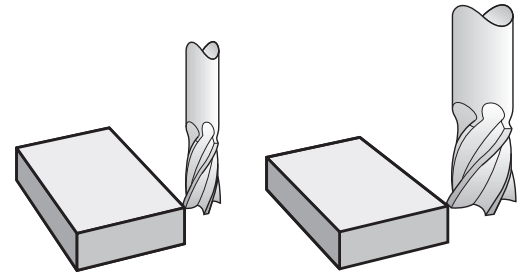
5.3 Tool compensation

Introduction

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

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

The TNC accounts for the compensation value in up to five axes including the rotary axes.



Tool length compensation

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



Danger of collision!

If you cancel a positive length compensation with **TOOL CALL 0** the distance between tool and workpiece will be reduced.

After **TOOL CALL** the path of the tool in the spindle axis, as entered in the part program, is adjusted by the difference between the length of the previous tool and that of the new one.

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

Compensation value = $L + DL_{TOOL CALL} + DL_{TAB}$ with

L: Tool length **L** from **TOOL DEF** block or tool table

DL_{TOOL CALL}: Oversize for length **DL** in the **TOOL CALL** block

DL_{TAB}: Oversize for length **DL** in the tool table

Tool radius compensation

The block for programming a tool movement contains:

- **RL** or **RR** for radius compensation
- **R0** 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 **RL** or **RR**.



The TNC automatically cancels radius compensation if you:

- program a straight line block with **R0**
- depart the contour with the **DEP** function
- program a **PGM CALL**
- Select a new program with **PGM MGT**

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

Compensation value = $R + DR_{\text{TOOL CALL}} + DR_{\text{TAB}}$ with

R: Tool radius **R** from **TOOL DEF** block or tool table

DR_{TOOL} Oversize for radius **DR** in the **TOOL CALL** block

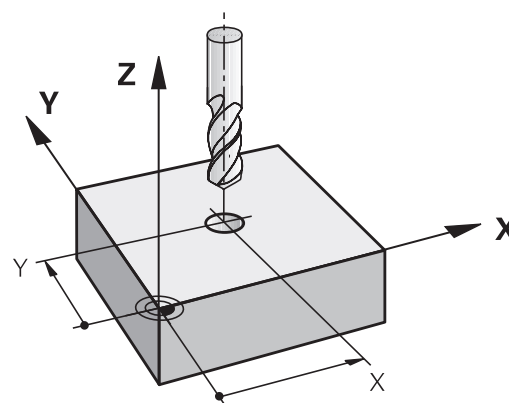
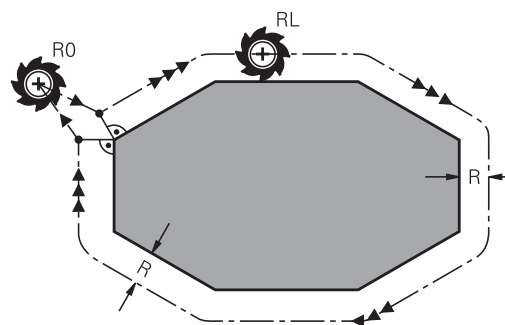
CALL:

DR_{TAB}: Oversize for radius **DR** in the tool table

Contouring without radius compensation: R0

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

Applications: Drilling and boring, pre-positioning



Programming: Tools

5.3 Tool compensation

Contouring with radius compensation: RR and RL

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

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

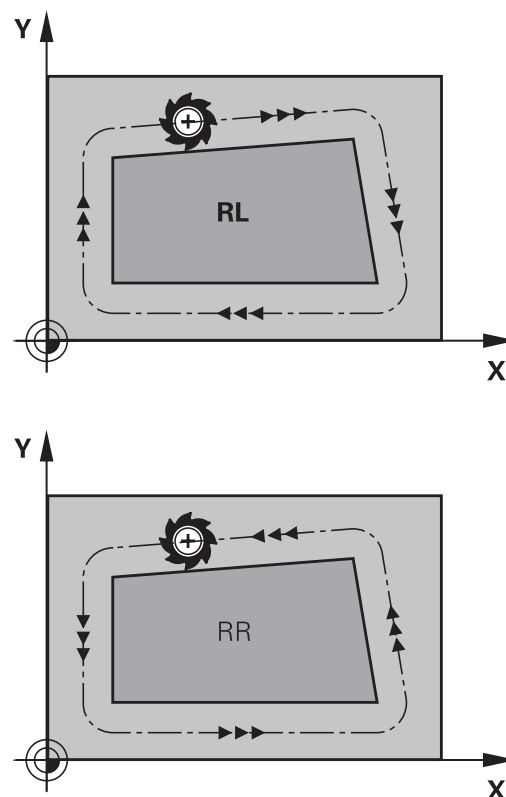
The tool center moves along the contour at a distance equal to the radius. "Right" or "left" are to be understood as based on the direction of tool movement along the workpiece contour. See figures.



Between two program blocks with different radius compensations **RR** and **RL** you must program at least one traversing block in the working plane without radius compensation (that is, with **R0**).

The TNC does not put radius compensation into effect until the end of the block in which it is first programmed.

In the first block in which radius compensation is activated with **RR/RL** or canceled with **R0** the TNC 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 an **L** block. Enter the coordinates of the target point and confirm your entry with ENT

RADIUS COMP.: RL/RR/NO COMP.?

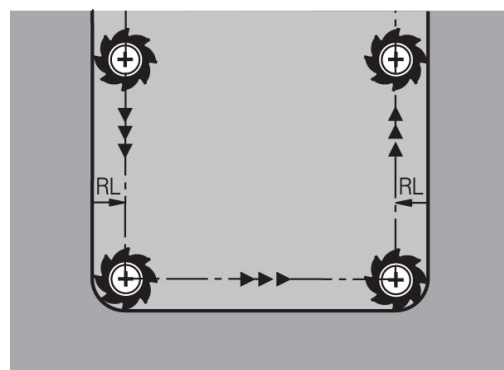
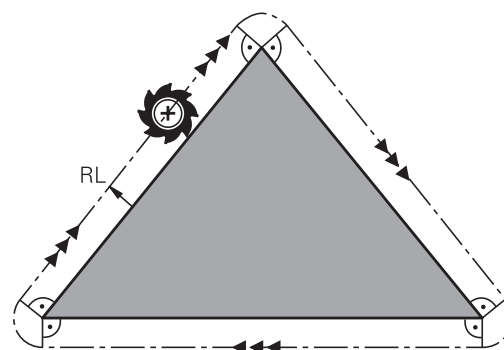
- | | |
|----------|--|
| RL | ▶ Select tool movement to the left of the contour:
Press the RL soft key, or |
| RR | ▶ Select tool movement to the right of the contour:
Press the RR soft key, or |
| ENT | ▶ Select tool movement without radius compensation or cancel radius compensation:
Press the ENT key |
| END
□ | ▶ Terminate the block: Press the END key |

Radius compensation: Machining corners

- **Outside corners:**
If you program radius compensation, the TNC moves the tool around outside corners on a transitional arc. If necessary, the TNC reduces the feed rate at outside corners to reduce machine stress, for example at very great changes of direction.
- **Inside corners:**
The TNC 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.

**Danger of collision!**

To prevent the tool from damaging the contour, be careful not to program the starting or end position for machining inside corners at a corner of the contour.



6

**Programming:
Programming
contours**

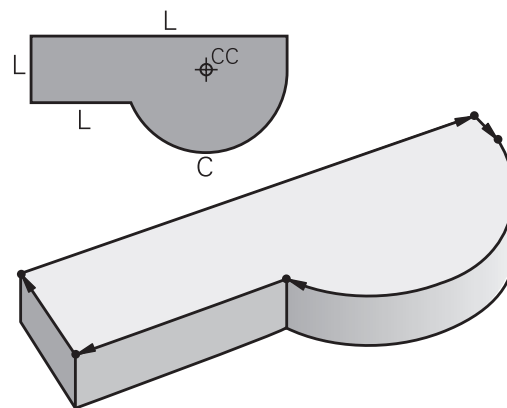
6 Programming: Programming contours

6.1 Tool movements

6.1 Tool movements

Path functions

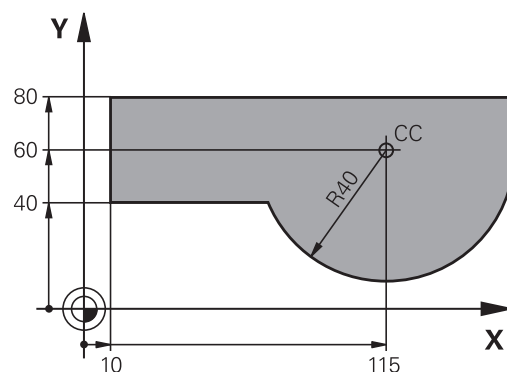
A workpiece contour is usually composed of several contour elements such as straight lines and circular arcs. With the path functions, you can program the tool movements for **straight lines** and **circular arcs**.



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 TNC calculates the missing data.

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



Miscellaneous functions M

With the TNC'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 program section only under certain conditions, you also define this machining sequence as a subprogram. In addition, you can have a part program call a separate program for execution.

Programming with subprograms and program section repeats: see "Programming: Subprograms and program section repeats", page 279.

Programming with Q parameters

Instead of programming numerical values in a part program, you enter markers called Q parameters. You assign the values to the Q parameters separately with the Q parameter functions. You can use the Q parameters for programming mathematical functions that control program execution or describe a contour.

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

Programming with Q parameters: see " Programming: Q Parameters", page 297.

Programming: Programming contours

6.2 Fundamentals of Path Functions

6.2 Fundamentals of Path Functions

Programming tool movements for workpiece machining

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

The TNC moves all axes programmed in a single block simultaneously.

Movement parallel to the machine axes

The program block contains only one coordinate. The TNC thus moves the tool parallel to the programmed axis.

Depending on the individual machine tool, the part program is executed by movement of either the tool or the machine table on which the workpiece is clamped. Nevertheless, you always program path contours as if the tool were moving and the workpiece remaining stationary.

Example:

```
50 L X+100
```

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

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

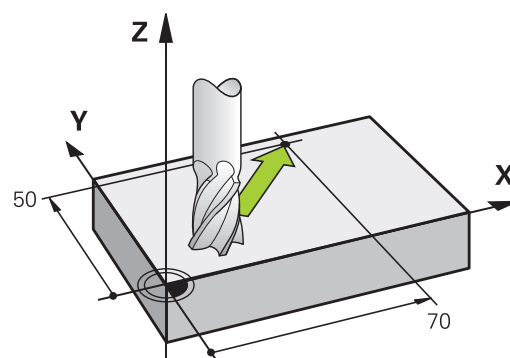
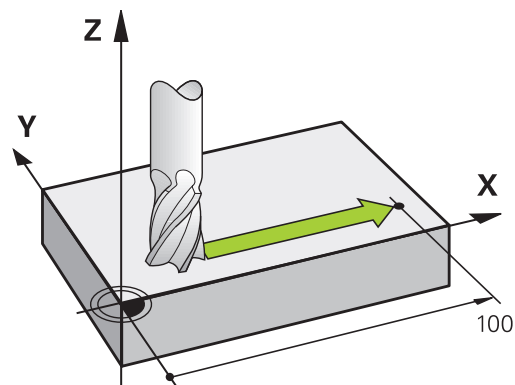
Movement in the main planes

The program block contains two coordinates. The TNC thus moves the tool in the programmed plane.

Example

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

The tool retains the Z coordinate and moves in the XY plane to the position X=70, Y=50 (see figure).

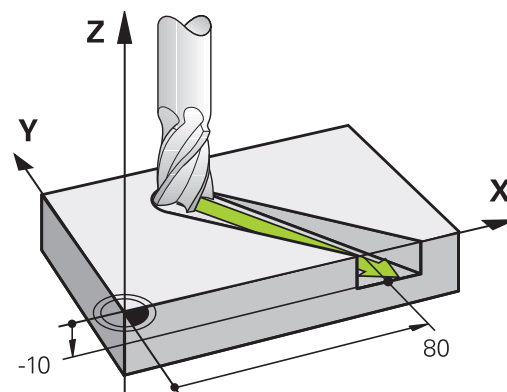


Three-dimensional movement

The program block contains three coordinates. The TNC thus moves the tool in space to the programmed position.

Example

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

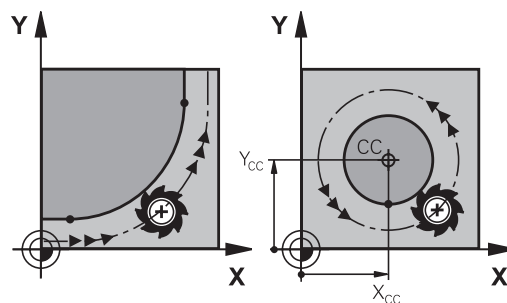


Circles and circular arcs

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

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

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



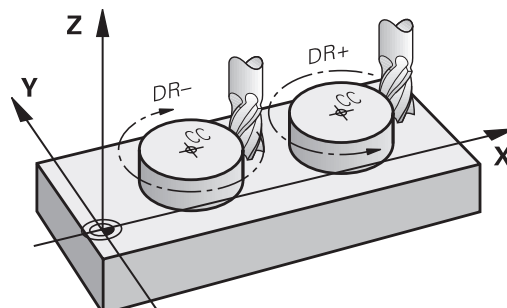
You can program circles that do not lie parallel to a main plane by using the function for tilting the working plane (see User's Manual for Cycles, Cycle 19, WORKING PLANE) or Q parameters (see "Principle and overview of functions", page 298).

Direction of rotation DR for circular movements

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

Clockwise direction of rotation: **DR-**

Counterclockwise direction of rotation: **DR+**



6

Programming: Programming contours

6.2 Fundamentals of Path Functions

Radius compensation

The radius compensation must be in the block in which you move to the first contour element. You cannot activate radius compensation in a circle block. It must be activated beforehand in a straight-line block (see "Path contours - Cartesian coordinates", page 222) or approach block (APPR block, see "Approaching and departing a contour", page 212).

Pre-position

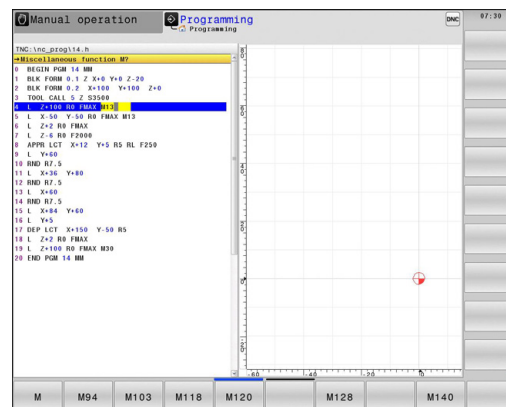


Danger of collision!

Before running a part program, always pre-position the tool to prevent the possibility of damaging it or the workpiece.

Creating the program blocks with the path function keys

The gray path function keys initiate the plain-language dialog. The TNC asks you successively for all the necessary information and inserts the program block into the part program.



Example – programming a straight line

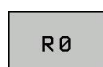
- ▶ Initiate the programming dialog, e.g. for a straight line

COORDINATES?

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

COORDINATES?

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

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

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

Feed rate F=? / F MAX = ENT

- ▶ Enter **100** (feed rate e.g. 100 mm/min), and confirm your entry with **ENT**. For programming in inches, enter 100 for a feed rate of 10 inches per minute. Or,



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



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

MISCELLANEOUS FUNCTION M?

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

The part program now contains the following line:

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


Programming: Programming contours

6.3 Approaching and departing a contour

6.3 Approaching and departing a contour

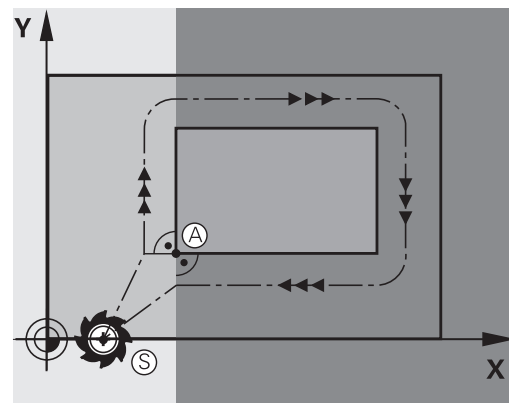
Starting point and end point

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

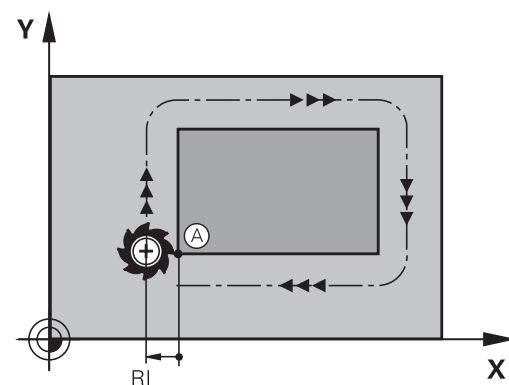
Figure at right:

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.



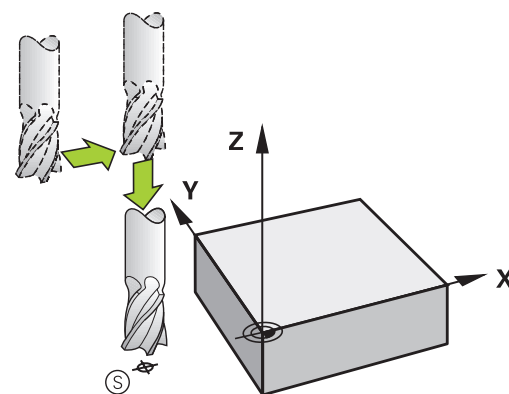
Approaching the starting point in the spindle axis

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.

NC blocks

```
30 L Z-10 R0 FMAX
```

```
31 L X+20 Y+30 RL F350
```



Approaching and departing a contour 6.3

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

Figure at right:

If you set the ending point in the dark gray area, the contour will be damaged when the end point is approached.

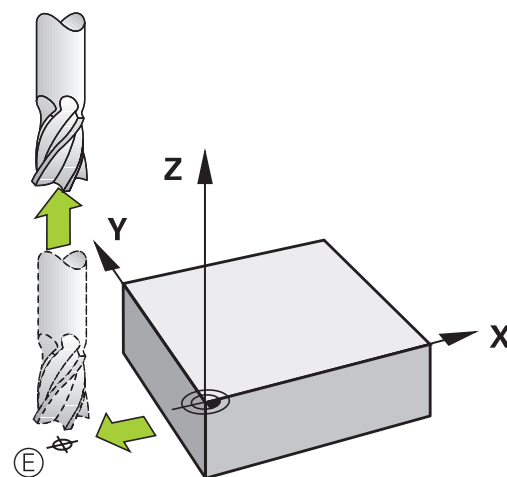
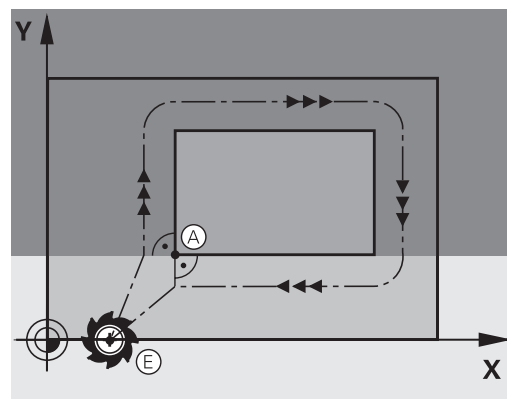
Departing the end point in the spindle axis:

Program the departure from the end point in the spindle axis separately. See figure at center right.

NC blocks

```
50 L X+60 Y+70 R0 F700
```

```
51 L Z+250 R0 FMAX
```



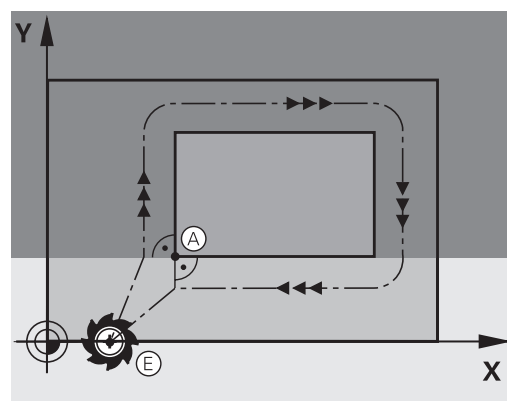
Common starting and end points

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

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

Figure at right:

If you set the ending point in the dark gray area, the contour will be damaged when the contour is approached / departed.











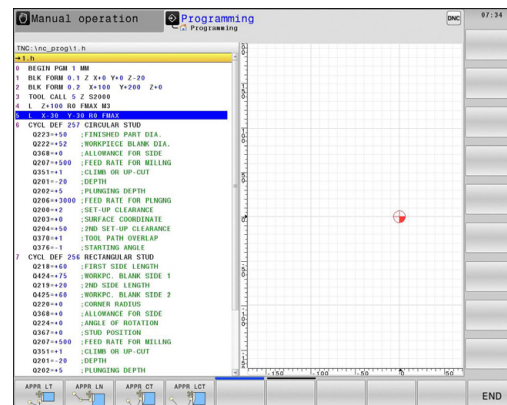
Programming: Programming contours

6.3 Approaching and departing a contour

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 desired path function with the corresponding soft key:

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



Important positions for approach and departure

- Starting point P_S

You program this position in the block before the APPR block. P_S lies outside the contour and is approached without radius compensation (R0).

- Auxiliary point P_H

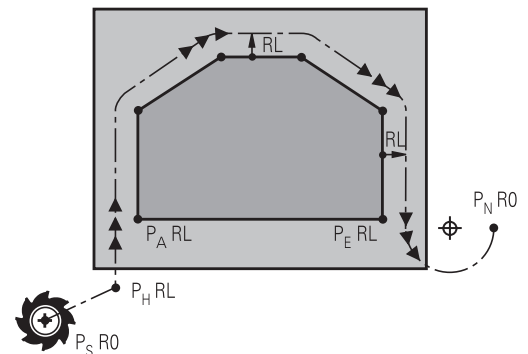
Some of the paths for approach and departure go through an auxiliary point P_H that the TNC calculates from your input in the APPR or DEP block. The TNC moves from the current position to the auxiliary point P_H at the feed rate last programmed. If you have programmed **FMAX** (positioning at rapid traverse) in the last positioning block before the approach function, the TNC also approaches the auxiliary point P_H at rapid traverse.

- 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 contains a Z axis coordinate, the TNC will first move the tool to P_H in the working plane, and then move it to the entered depth in the tool axis.

- 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 contains a Z axis coordinate, the TNC will first move the tool to P_N in the working plane, and then move it to the entered height in the tool axis.



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



The TNC does not check whether the programmed contour will be damaged when moving from the actual position to the auxiliary point P_H . Use the test graphics to check.

With the APPR LT, APPR LN and APPR CT functions, the TNC moves the tool from the actual position to the auxiliary point P_H at the feed rate/rapid traverse that was last programmed. With the APPR LCT function, the TNC moves to the auxiliary point P_H at the feed rate programmed with the APPR block. If no feed rate is programmed before the approach block, the TNC generates an error message.

Programming: Programming contours

6.3 Approaching and departing a contour

Polar coordinates

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

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

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

Radius compensation

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



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

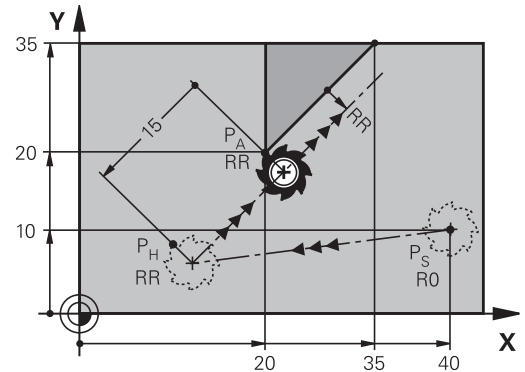
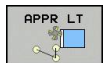
This method of function differs from the iTNC 530 control!

Approaching and departing a contour 6.3

Approaching on a straight line with tangential connection: APPR LT

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

- ▶ Use any path function to approach the starting point P_S .
- ▶ Initiate the dialog with the **APPR/DEP** key and **APPR LT** soft key:
 - ▶ Coordinates of the first contour point P_A
 - ▶ **LEN**: Distance from the auxiliary point P_H to the first contour point P_A
 - ▶ Radius compensation **RR/RL** for machining

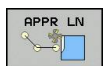


Example NC blocks

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

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

- ▶ Use any path function to approach the starting point P_S .
- ▶ Initiate the dialog with the **APPR/DEP** key and **APPR LN** soft key:
 - ▶ Coordinates of the first contour point P_A
 - ▶ Length: Distance to the auxiliary point P_H . Always enter **LEN** as a positive value!
 - ▶ Radius compensation **RR/RL** for machining



Example NC blocks

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

Programming: Programming contours

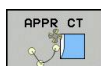
6.3 Approaching and departing a contour

Approaching on a circular path with tangential connection: APPR CT

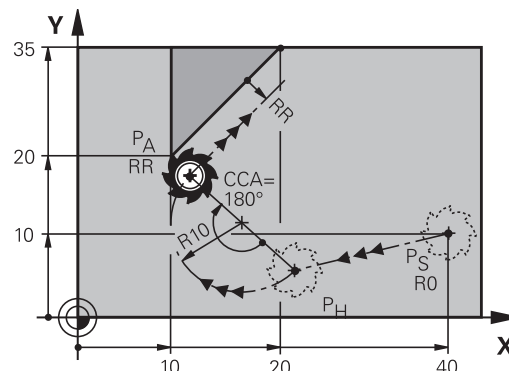
The tool moves on a straight line from the starting point P_S to an auxiliary point P_H . It then moves from P_H to the first contour point P_A following a circular arc that is tangential to the first contour element.

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 from the workpiece side: Enter R as a negative value.
- ▶ Center angle **CCA** of the arc
 - CCA can be entered only as a positive value.
 - Maximum input value 360°
- ▶ Radius compensation **RR/RL** for machining



Example NC blocks

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

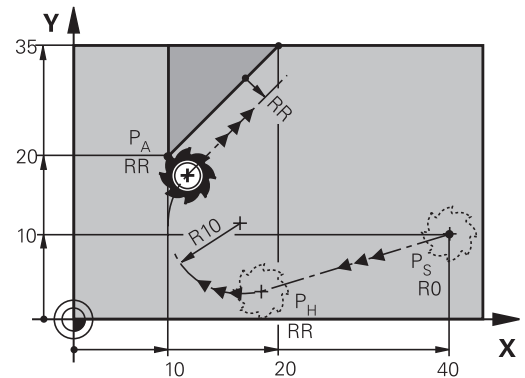
Approaching on a circular path with tangential connection from a straight line to the contour: APPR LCT

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

If you program all three principal axes X, Y and Z in the approach block, the TNC initially traverses the tool from the starting point P_S in the working plane, and then in the tool axis on the auxiliary point P_H . The control only traverses the tool in the working plane from auxiliary point P_H to the contour point P_A .

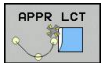


Consider this behavior when importing programs from earlier controls. Adapt the program if required. Earlier controls traverse the auxiliary point P_H in all three principal axes simultaneously.



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

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



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

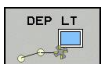
Example NC blocks

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

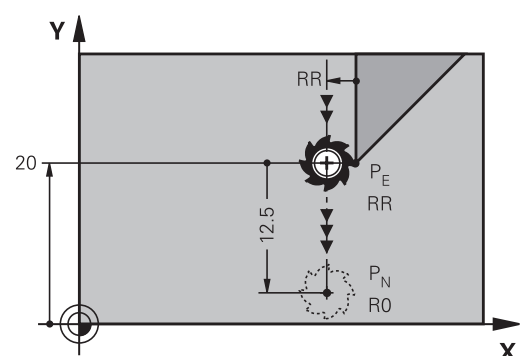
Departing in a straight line with tangential connection: DEP LT

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

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



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



6 Programming: Programming contours

6.3 Approaching and departing a contour

Example NC blocks

23 L Y+20 RR F100	Last contour element: PE with radius compensation
24 DEP LT LEN12.5 F100	Depart contour by LEN=12.5 mm
25 L Z+100 FMAX M2	Retract in Z, return to block 1, end program

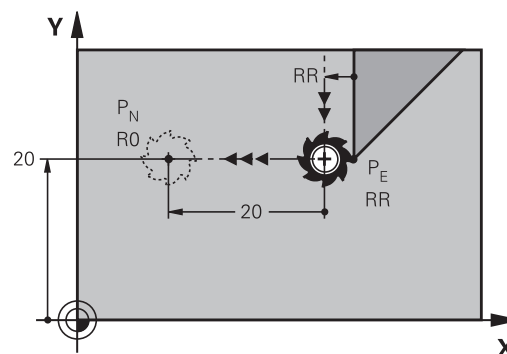
Departing in a straight line perpendicular to the last contour point: DEP LN

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 of the end point P_N .
Remember: always enter **LEN** as a positive value!



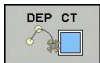
Example NC blocks

23 L Y+20 RR F100	Last contour element: PE with radius compensation
24 DEP LN LEN+20 F100	Depart perpendicular to contour by LEN=20 mm
25 L Z+100 FMAX M2	Retract in Z, return to block 1, end program

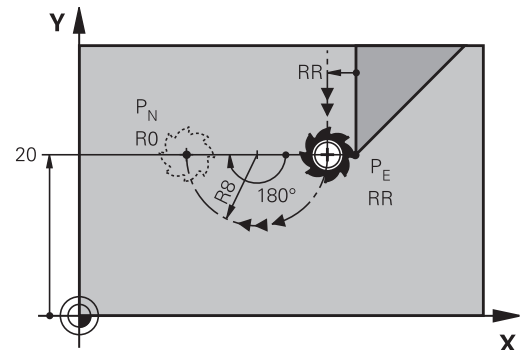
Departing on a circular path with tangential connection: DEP CT

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

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



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



Example NC blocks

23 L Y+20 RR F100	Last contour element: PE with radius compensation
24 DEP CT CCA 180 R+8 F100	Center angle=180°, arc radius=8 mm
25 L Z+100 FMAX M2	Retract in Z, return to block 1, end program

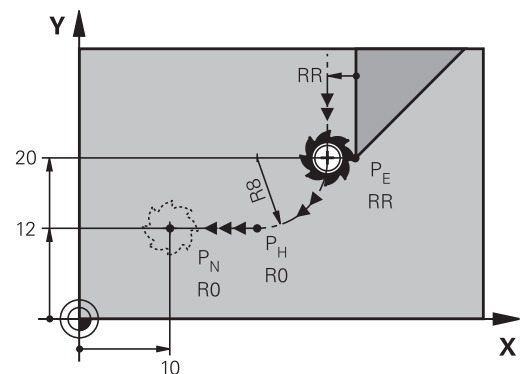
Departing on a circular arc tangentially connecting the contour and a straight line: DEP LCT

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

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



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



Example NC blocks









23 L Y+20 RR F100	Last contour element: PE with radius compensation
24 DEP LCT X+10 Y+12 R+8 F100	Coordinates PN, arc radius=8 mm
25 L Z+100 FMAX M2	Retract in Z, return to block 1, end program

Programming: Programming contours

6.4 Path contours - Cartesian coordinates

6.4 Path contours - Cartesian coordinates

Overview of path functions

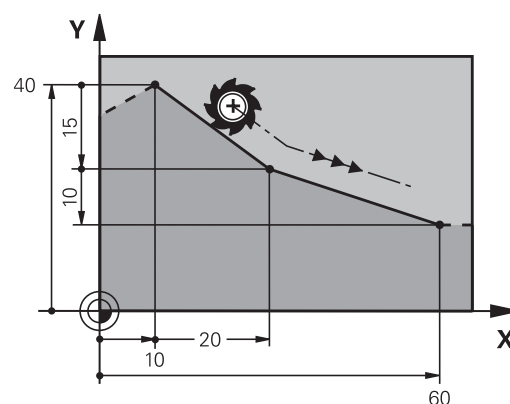
Path function key	Function	Tool movement	Required input	Page
	Straight line L	Straight line	Coordinates of the end point of the straight line	223
	Chamfer: CHF	Chamfer between two straight lines	Chamfer side length	224
	Circle center CC	None	Coordinates of the circle center or pole	226
	Circular arc C	Circular arc around a circle center CC to an arc end point	Coordinates of the arc end point, direction of rotation	227
	Circular arc CR	Circular arc with a certain radius	Coordinates of the arc end point, arc radius, direction of rotation	228
	Kreisbogen CT	Circular arc with tangential connection to the preceding and subsequent contour elements	Coordinates of the arc end point	230
	Corner rounding RND	Circular arc with tangential connection to the preceding and subsequent contour elements	Rounding radius R	225
	FK free contour programming	Straight line or circular path with any connection to the preceding contour element	see "Path contours – FK free contour programming", page 241	244

Straight line L

The TNC 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 block.



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



Example NC blocks

```
7 L X+10 Y+40 RL F200 M3
```

```
8 L IX+20 IY-15
```

```
9 L X+60 IY-10
```

Capture actual position

You can also generate a straight-line block (**L** 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 and Editing
- ▶ Select the program block after which you want to insert the L block



- ▶ Press the **ACTUAL-POSITION-CAPTURE** key: The TNC generates an L block with the actual position coordinates.

Programming: Programming contours

6.4 Path contours - Cartesian coordinates

Inserting a chamfer between two straight lines

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

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



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

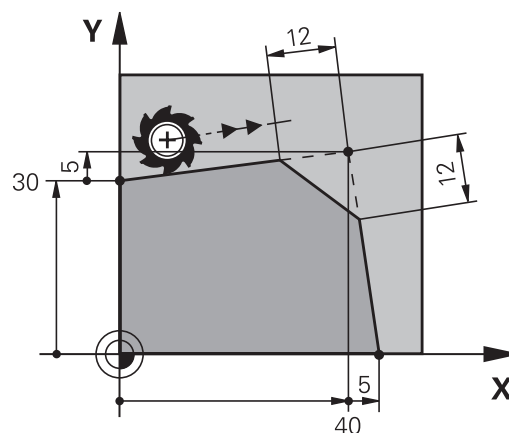
Example NC blocks

```
7 L X+0 Y+30 RL F300 M3
```

```
8 L X+40 IY+5
```

```
9 CHF 12 F250
```

```
10 L IX+5 Y+0
```



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

Corner rounding RND

The **RND** function is used for rounding off corners.

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

The rounding arc must be machinable with the called tool.



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

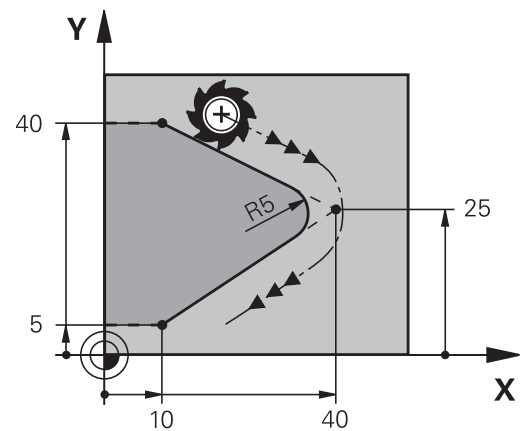
Example NC blocks

```
5 L X+10 Y+40 RL F300 M3
```

```
6 L X+40 Y+25
```

```
7 RND R5 F100
```

```
8 L X+10 Y+5
```



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

The corner point is cut off by the rounding arc and is not part of the contour.

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

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

Programming: Programming contours

6.4 Path contours - Cartesian coordinates

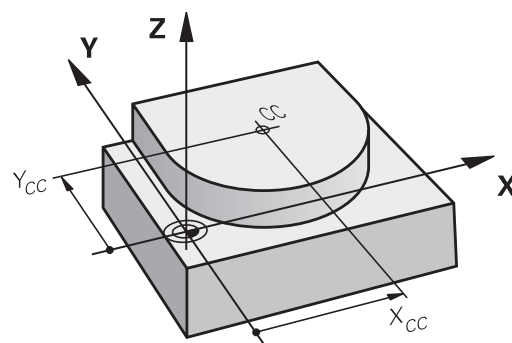
Circle center CC

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

- Entering the Cartesian coordinates of the circle center in the working plane, or
- Using the circle center defined in an earlier block, or
- Capturing the coordinates with the **ACTUAL-POSITION-CAPTURE** key



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



Example NC blocks

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

or

```
10 L X+25 Y+25
```

```
11 CC
```

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

Validity

The circle center definition remains in effect until a new circle center is programmed.

Entering the circle center incrementally

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



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

Circular path C around circle center CC

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

- Move the tool to the circle starting point



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



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



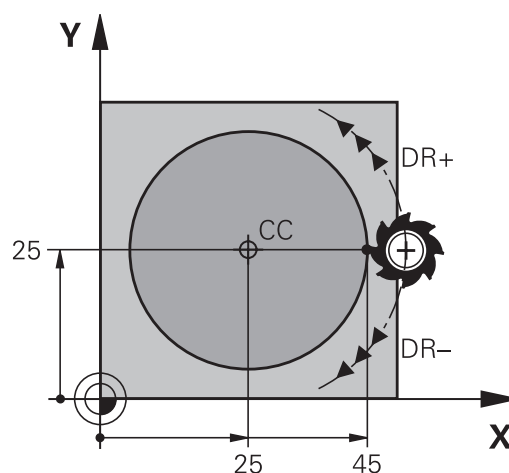
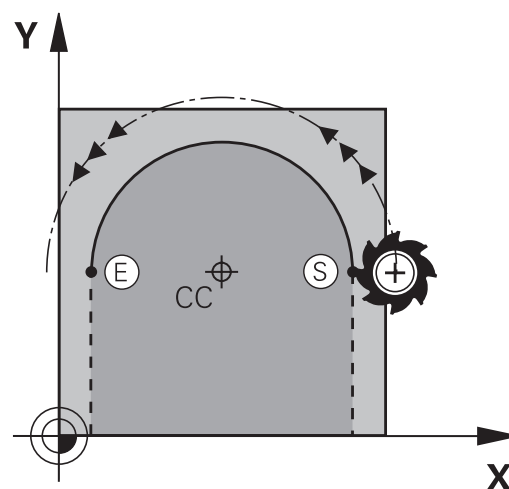
The TNC normally makes circular movements in the active working plane. If you program circular arcs that do not lie in the active working plane, e.g. **C Z... X... DR+** with a tool axis Z, and at the same time rotate this movement, then the TNC moves the tool in a spatial arc, which means a circular arc in 3 axes (software option 8).

Example NC blocks

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

```
6 L X+45 Y+25 RR F200 M3
```

```
7 C X+45 Y+25 DR+
```



Full circle

For the end point, enter the same point that you used for the starting point.



The starting and end points of the arc must lie on the circle.

Input tolerance: up to 0.016 mm (selected through the **circleDeviation** machine parameter).

Smallest possible circle that the TNC can traverse: 0.0016 µm.

Programming: Programming contours

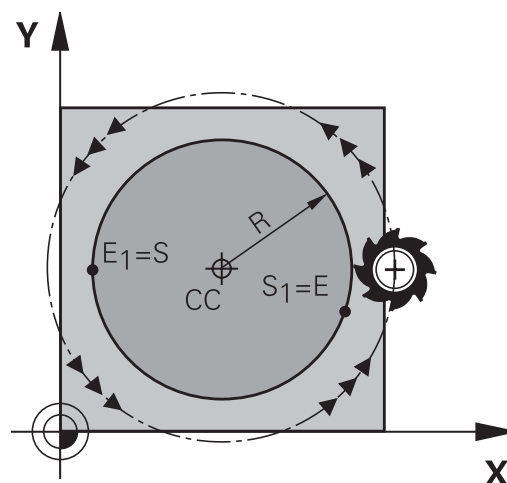
6.4 Path contours - Cartesian coordinates

CircleCR with defined radius

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



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



Full circle

For a full circle, program two 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: $CCA < 180^\circ$

Enter the radius with a positive sign $R > 0$

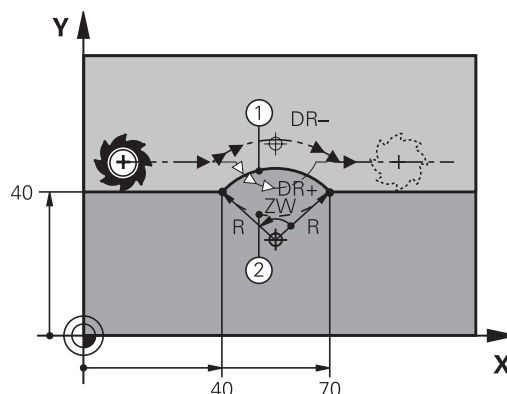
Larger arc: $CCA > 180^\circ$

Enter the radius with a negative sign $R < 0$

The direction of rotation determines whether the arc is curving outward (convex) or curving inward (concave):

Convex: Direction of rotation **DR-** (with radius compensation **RL**)

Concave: Direction of rotation **DR+** (with radius compensation **RL**)



The distance from the starting and end points of the arc diameter cannot be greater than the diameter of the arc.

The maximum radius is 99.9999 m.

You can also enter rotary axes A, B and C.

Example NC blocks

```
10 L X+40 Y+40 RL F200 M3
```

```
11 CR X+70 Y+40 R+20 DR- (ARC 1)
```

or

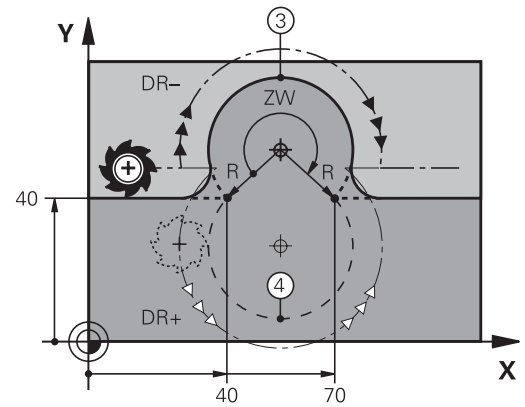
```
11 CR X+70 Y+40 R+20 DR+ (ARC 2)
```

or

```
11 CR X+70 Y+40 R-20 DR- (ARC 3)
```

or

```
11 CR X+70 Y+40 R-20 DR+ (ARC 4)
```



Programming: Programming contours

6.4 Path contours - Cartesian coordinates

Circle CT with tangential connection

The tool moves on an arc that starts tangentially to the previously programmed contour element.

A transition between two contour elements is called tangential when there is no kink or corner at the intersection between the two contours—the transition is smooth.

The contour element to which the tangential arc connects must be programmed immediately before the **CT** block. This requires at least two positioning blocks.



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

Example NC blocks

```
7 L X+0 Y+25 RL F300 M3
```

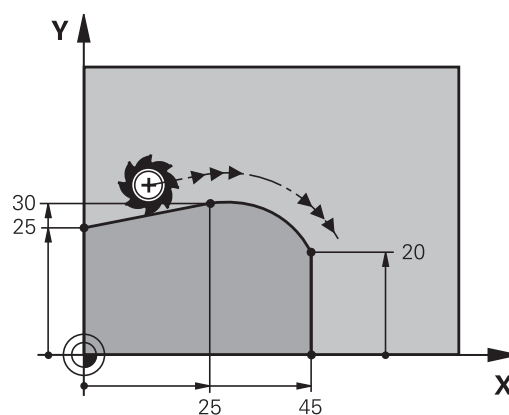
```
8 L X+25 Y+30
```

```
9 CT X+45 Y+20
```

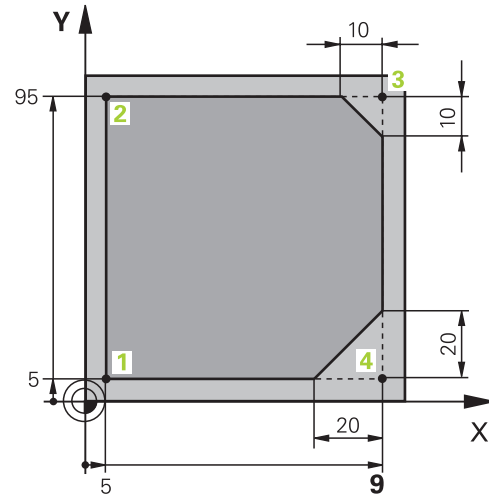
```
10 L Y+0
```



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



Example: Linear movements and chamfers with Cartesian coordinates

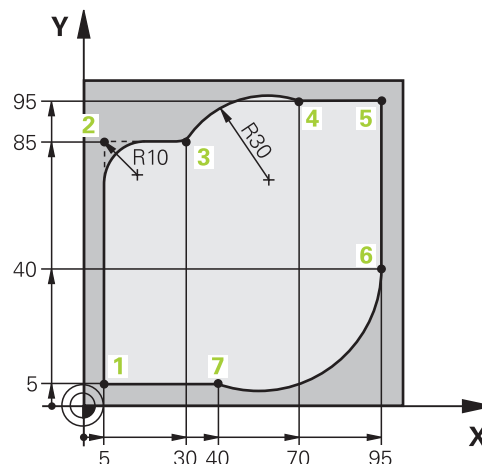


0 BEGIN PGM LINEAR MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	Define the workpiece blank for graphic workpiece simulation
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 1 Z S4000	Call the tool in the spindle axis and with spindle speed
4 L Z+250 R0 FMAX	Retract the tool in the spindle axis at rapid traverse FMAX
5 L X-10 Y-10 R0 FMAX	Pre-position the tool
6 L Z-5 R0 F1000 M3	Move to working depth at feed rate F = 1000 mm/min
7 APPR LT X+5 Y+5 LEN10 RL F300	Approach the contour at point 1 on a straight line with tangential connection
8 L Y+95	Move to point 2
9 L X+95	Point 3: first straight line for corner 3
10 CHF 10	Program a chamfer with length 10 mm
11 L Y+5	Point 4: 2nd straight line for corner 3, 1st straight line for corner 4
12 CHF 20	Program a chamfer with length 20 mm
13 L X+5	Move to last contour point 1, second straight line for corner 4
14 DEP LT LEN10 F1000	Depart the contour on a straight line with tangential connection
15 L Z+250 R0 FMAX M2	Retract the tool, end program
16 END PGM LINEAR MM	

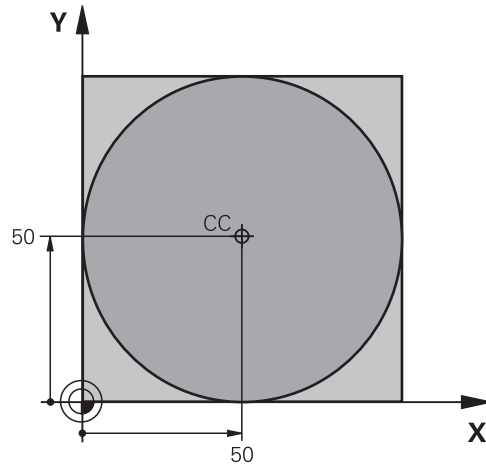
Programming: Programming contours

6.4 Path contours - Cartesian coordinates

Example: Circular movements with Cartesian coordinates



0 BEGIN PGM CIRCULAR MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	Define the workpiece blank for graphic workpiece simulation
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 1 Z S4000	Call the tool in the spindle axis and with spindle speed
4 L Z+250 R0 FMAX	Retract the tool in the spindle axis at rapid traverse FMAX
5 L X-10 Y-10 R0 FMAX	Pre-position the tool
6 L Z-5 R0 F1000 M3	Move to working depth at feed rate F = 1000 mm/min
7 APPR LCT X+5 Y+5 R5 RL F300	Approach the contour at point 1 on a circular arc with tangential connection
8 L X+5 Y+85	Point 2: First straight line for corner 2
9 RND R10 F150	Insert radius with R = 10 mm, feed rate: 150 mm/min
10 L X+30 Y+85	Move to point 3: Starting point of the arc with CR
11 CR X+70 Y+95 R+30 DR-	Move to point 4: End point of the arc with CR, radius 30 mm
12 L X+95	Move to point 5
13 L X+95 Y+40	Move to point 6
14 CT X+40 Y+5	Move to point 7: End point of the arc, circular arc with tangential connection to point 6, TNC automatically calculates the radius
15 L X+5	Move to last contour point 1
16 DEP LCT X-20 Y-20 R5 F1000	Depart the contour on a circular arc with tangential connection
17 L Z+250 R0 FMAX M2	Retract the tool, end program
18 END PGM CIRCULAR MM	

Example: Full circle with Cartesian coordinates

0 BEGIN PGM C-CC MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	Definition of workpiece blank
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 1 Z S3150	Tool call
4 CC X+50 Y+50	Define the circle center
5 L Z+250 R0 FMAX	Retract the tool
6 L X-40 Y+50 R0 FMAX	Pre-position the tool
7 L Z-5 R0 F1000 M3	Move to working depth
8 APPR LCT X+0 Y+50 R5 RL F300	Approach the starting point of the circle on a circular arc with tangential connection
9 C X+0 DR-	Move to the circle end point (= circle starting point)
10 DEP LCT X-40 Y+50 R5 F1000	Depart the contour on a circular arc with tangential connection
11 L Z+250 R0 FMAX M2	Retract the tool, end program
12 END PGM C-CC MM	

Programming: Programming contours

6.5 Path contours – Polar coordinates

6.5 Path contours – Polar coordinates









Overview

With polar coordinates you can define a position in terms of its angle **PA** and its distance **PR** relative to a previously defined pole **CC**.

Polar coordinates are useful with:

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

Overview of path functions with polar coordinates

Path function key	Tool movement	Required input	Page
 + 	Straight line	Polar radius, polar angle of the straight-line end point	235
 + 	Circular path around circle center/pole to arc end point	Polar angle of the arc end point, direction of rotation	236
 + 	Circular arc with tangential connection to the preceding contour element	Polar radius, polar angle of the arc end point	236
 + 	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	237

Zero point for polar coordinates: pole CC

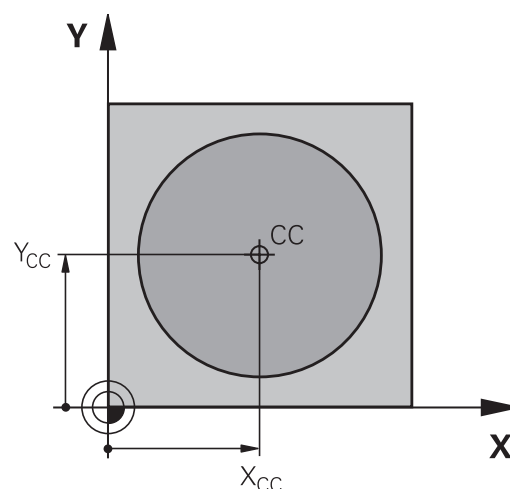
You can set the pole CC at any point in the machining program, before indicating points in polar coordinates. Set the pole in the same way as you would program the circle center.



- **Coordinates:** Enter Cartesian coordinates for the pole or, if you want to use the last programmed position, do not enter any coordinates. Before programming polar coordinates, define the pole. You can only define the pole in Cartesian coordinates. The pole remains in effect until you define a new pole.

Example NC blocks

```
12 CC X+45 Y+25
```



Straight line LP

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



- **Polar coordinate radius PR:** Enter the distance from the pole CC to the straight-line end point.



- **Polar coordinate angle PA:** Angular position of the straight-line end point between -360° and $+360^\circ$

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

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

Example NC blocks

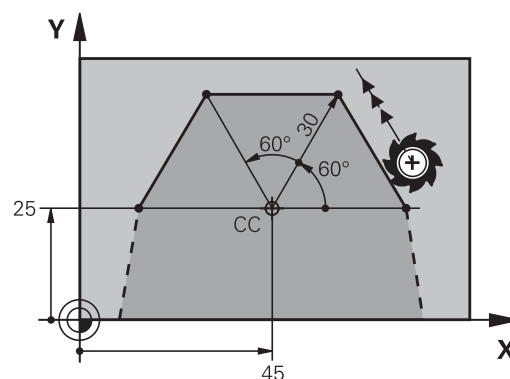
```
12 CC X+45 Y+25
```

```
13 LP PR+30 PA+0 RR F300 M3
```

```
14 LP PA+60
```

```
15 LP IPA+60
```

```
16 LP PA+180
```



Programming: Programming contours

6.5 Path contours – Polar coordinates

Circular path CP around pole CC

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



- **Polar coordinate angle PA:** Angular position of the arc end point between -99999.9999° and $+99999.9999^\circ$



- **Direction of rotation DR**

Example NC blocks

```
18 CC X+25 Y+25
```

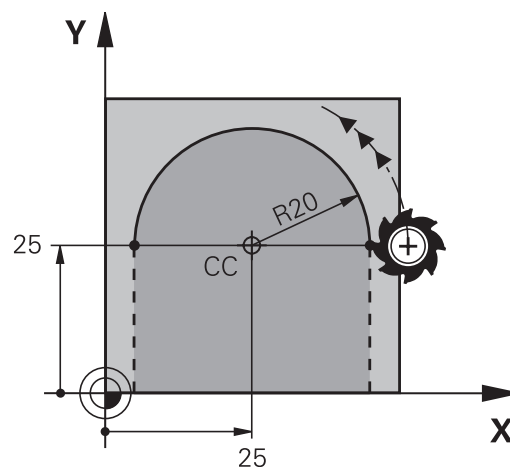
```
19 LP PR+20 PA+0 RR F250 M3
```

```
20 CP PA+180 DR+
```



For incremental coordinates, enter the same sign for DR and PA.

Consider this behavior when importing programs from earlier controls. Adapt the program if required.



Circle CTP with tangential connection

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



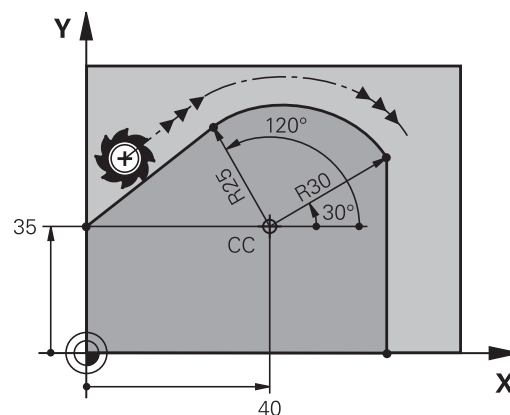
- **Polar coordinate radius PR:** Distance between the arc end point and the pole **CC**



- **Polar coordinate angle PA:** Angular position of the arc end point.



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



Example NC blocks

```
12 CC X+40 Y+35
```

```
13 L X+0 Y+35 RL F250 M3
```

```
14 LP PR+25 PA+120
```

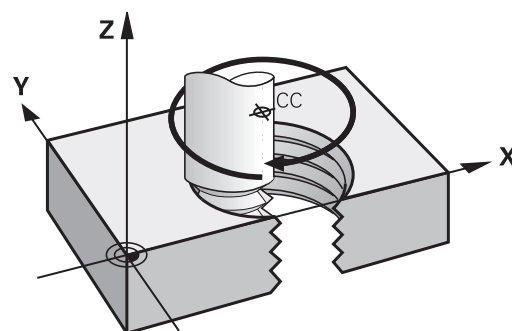
```
15 CTP PR+30 PA+30
```

```
16 L Y+0
```


Helix

A helix is a combination of a circular movement in a main plane and a linear movement perpendicular to this plane. You program the circular path in a main plane.

A helix is programmed only in polar coordinates.



Application

- Large-diameter internal and external threads
- Lubrication grooves

Calculating the helix

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

Thread revolutions n:	Thread revolutions + overrun at start and end of thread
Total height h:	Thread pitch P times thread revolutions n
Incremental total angle IPA:	Thread revolutions x 360° + angle for beginning of thread + angle for thread overrun
Starting coordinate Z:	Pitch P times (thread revolutions + thread overrun at start of thread)

Shape of the helix

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

Internal thread	Work direction	Direction of rotation	Radius compensation
Right-hand	Z+	DR+	RL
Left-hand	Z+	DR-	RR
Right-hand	Z-	DR-	RR
Left-hand	Z-	DR+	RL
External thread			
Right-hand	Z+	DR+	RR
Left-hand	Z+	DR-	RL
Right-hand	Z-	DR-	RL
Left-hand	Z-	DR+	RR

Programming: Programming contours

6.5 Path contours – Polar coordinates

Programming a helix

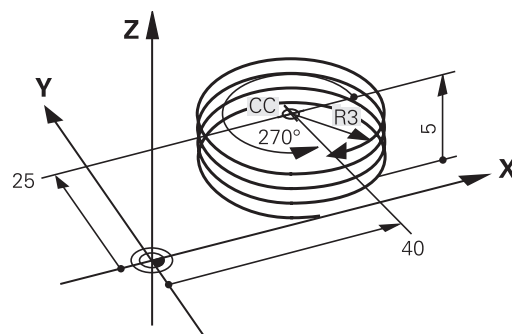


Always enter the same algebraic sign for the direction of rotation and the incremental total angle **IPA**. The tool may otherwise move in a wrong path and damage the contour.

For the total angle **IPA** you can enter a value of $-99\,999.9999^\circ$ to $+99\,999.9999^\circ$.



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



Example NC blocks: Thread M6 x 1 mm with 5 revolutions

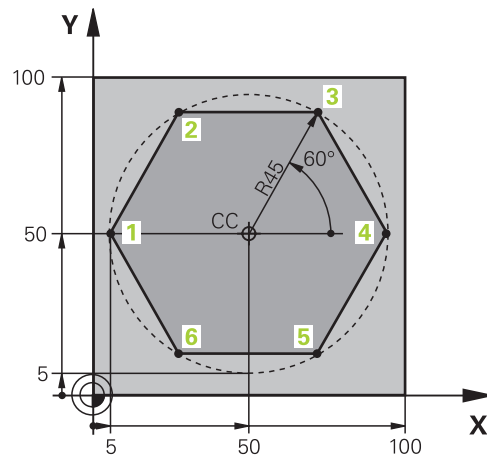
12 CC X+40 Y+25

13 L Z+0 F100 M3

14 LP PR+3 PA+270 RL F50

15 CP IPA-1800 IZ+5 DR-

Example: Linear movement with polar coordinates

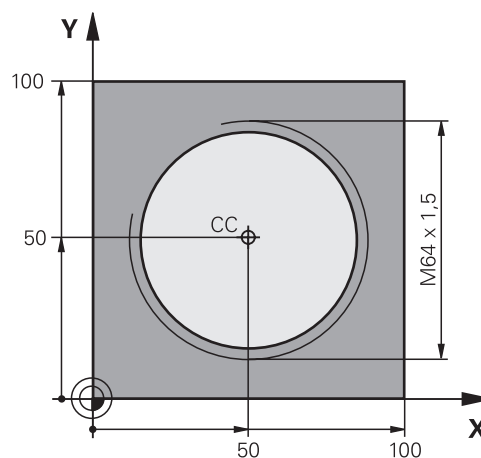


0 BEGIN PGM LINEARPO MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	Definition of workpiece blank
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 1 Z S4000	Tool call
4 CC X+50 Y+50	Define the datum for polar coordinates
5 L Z+250 R0 FMAX	Retract the tool
6 LP PR+60 PA+180 R0 FMAX	Pre-position the tool
7 L Z-5 R0 F1000 M3	Move to working depth
8 APPR PLCT PR+45 PA+180 R5 RL F250	Approach the contour at point 1 on a circular arc with tangential connection
9 LP PA+120	Move to point 2
10 LP PA+60	Move to point 3
11 LP PA+0	Move to point 4
12 LP PA-60	Move to point 5
13 LP PA-120	Move to point 6
14 LP PA+180	Move to point 1
15 DEP PLCT PR+60 PA+180 R5 F1000	Depart the contour on a circular arc with tangential connection
16 L Z+250 R0 FMAX M2	Retract the tool, end program
17 END PGM LINEARPO MM	

Programming: Programming contours

6.5 Path contours – Polar coordinates

Example: Helix



0 BEGIN PGM HELIX MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	Definition of workpiece blank
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 1 Z S1400	Tool call
4 L Z+250 R0 FMAX	Retract the tool
5 L X+50 Y+50 R0 FMAX	Pre-position the tool
6 CC	Transfer the last programmed position as the pole
7 L Z-12,75 R0 F1000 M3	Move to working depth
8 APPR PCT PR+32 PA-182 CCA180 R+2 RL F100	Approach the contour on a circular arc with tangential connection
9 CP IPA+3240 IZ+13,5 DR+ F200	Helical interpolation
10 DEP CT CCA180 R+2	Depart the contour on a circular arc with tangential connection
11 L Z+250 R0 FMAX M2	Retract the tool, end program
12 END PGM HELIX MM	

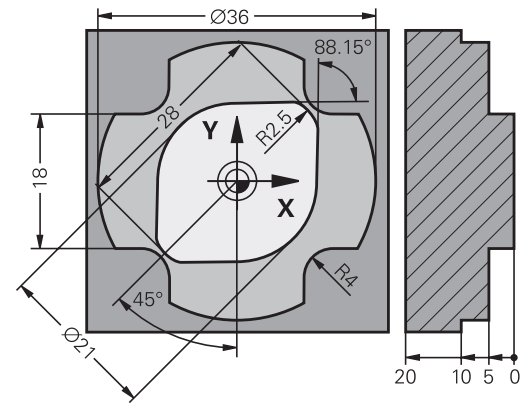
6.6 Path contours – FK free contour programming

Fundamentals

Workpiece drawings that are not dimensioned for NC often contain unconventional coordinate data that cannot be entered with the gray path function keys. For example:

- Known coordinates on the contour element or in its proximity
- Coordinate data can be referenced to another contour element
- Directional data and data regarding the course of the contour

You can enter such dimensional data directly by using the FK free contour programming function. The TNC derives the contour from the known coordinate data and supports the programming dialog with the interactive programming graphics. The figure at upper right shows a workpiece drawing for which FK programming is the most convenient programming method.



Programming: Programming contours

6.6 Path contours – FK free contour programming



The following prerequisites for FK programming must be observed:

The FK free contour programming feature can only be used for programming contour elements that lie in the working plane.

The working plane for FK programming is defined 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 run in turning mode
- 3. By the working plane pre-defined in the **TOOL CALL** (e.g. **TOOL CALL 1 TOOL CALLZ = X/Y plane**)
- 4. The standard X/Y plane is active if none of these applies

The display of the FK soft keys depends on the spindle axis in the workpiece blank definition. If for example you enter spindle axis **Z** in the workpiece blank definition, the TNC only shows FK soft keys for the X/Y plane.

You must enter all available data for every contour element. Even the data that does not change must be entered in every 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**), i.e. elements that are referenced to other NC blocks.

If both FK blocks and conventional blocks are entered in a program, the FK contour must be fully defined before you can return to conventional programming.

The TNC needs a fixed point from which it can calculate the contour elements. 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 block.

If the first 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 to fully define the direction of contour approach.

Do not program an FK contour immediately after an **LBL** command.

FK programming graphics



If you wish to use graphic support during FK programming, select the PROGRAM + GRAPHICS screen layout, see "Programming", page 76

Incomplete coordinate data often is not sufficient to fully define a workpiece contour. In this case, the TNC indicates the possible solutions in the FK graphic. You can then select the contour that matches the drawing. The FK graphic displays the elements of the workpiece contour in different colors:

- Blue:** The contour element is fully defined
The last FK element is only shown in blue after the departure movement, despite full definition, e.g. via CLSD-.
- Green:** The entered data describe a limited number of possible solutions: select the correct one
- Red:** The entered data are not sufficient to determine the contour element: enter further data

If the entered data permit a limited number of 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 (2nd soft-key row) if you cannot distinguish possible solutions in the standard setting

SELECT

SOLUTION

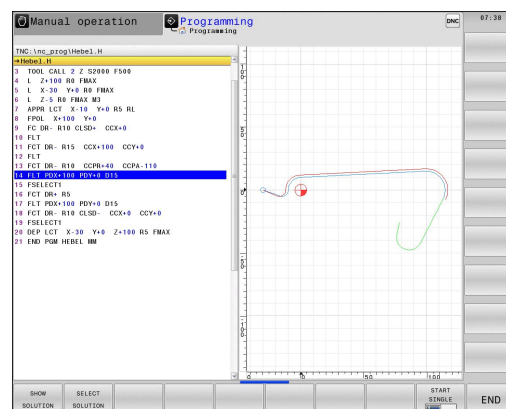
- ▶ If the displayed contour element matches the drawing, select the contour element with **SELECT SOLUTION**

If you do not yet wish to select a green contour element, press the **END SELECT** 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.

The machine tool builder may use other colors for the FK graphics.



Showing block numbers in the graphic window

To show a block number in the graphic window:

SHOW

OMIT

BLOCK NO.

- ▶ Set the **SHOW OMIT BLOCK NR.** soft key to **SHOW** (soft-key row 3)


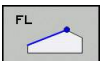
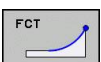

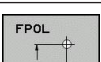
Programming: Programming contours

6.6 Path contours – FK free contour programming



Initiating the FK dialog

If you press the gray FK button, the TNC displays the soft keys you can use to initiate an FK dialog—see the following table. Press the **FK** button a second time to deselect the soft keys.

If you initiate the FK dialog with one of these soft keys, the TNC shows additional soft-key rows that you can use for entering 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

Pole for FK programming

- 
 - ▶ 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 TNC then displays the axis soft keys of the active working plane
 - ▶ Enter the pole coordinates using these soft keys



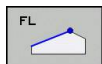
The pole for FK programming remains active until you define a new one using FPOL.

Free straight line programming

Straight line without tangential connection



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



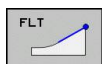
- ▶ To initiate the dialog for free programming of straight lines, press the **FL** soft key. The TNC displays additional soft keys
- ▶ Enter all known data in the block by using these soft keys. The FK graphic displays the programmed contour element in red until sufficient data is entered. If the entered data describes several solutions, the graphic will display the contour element in green (see "FK programming graphics", page 243)

Straight line with tangential connection

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



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



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

Programming: Programming contours

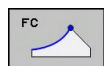
6.6 Path contours – FK free contour programming

Free circular path programming

Circular arc without tangential connection



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



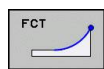
- ▶ To initiate the dialog for free programming of circular arcs, press the **FC** soft key. The TNC displays soft keys with which you can directly enter data on the circular arc or the circle center
- ▶ Enter all known data in the block by using these soft keys. The FK graphic displays the programmed contour element in red until sufficient data is entered. If the entered data describes several solutions, the graphic will display the contour element in green (see "FK programming graphics", page 243)

Circular arc with tangential connection

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



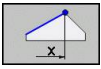
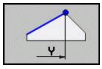
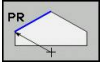
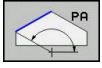
- ▶ To display the soft keys for free contour programming, press the **FK** key



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

Input options

End point coordinates

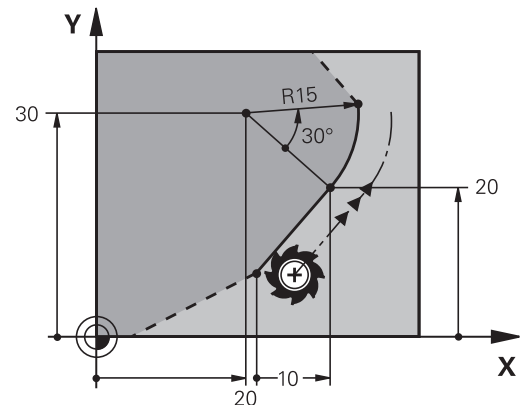
Soft keys	Known data
 	Cartesian coordinates X and Y
 	Polar coordinates referenced to FPOL

Example NC blocks

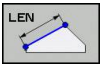
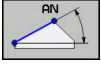


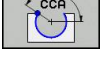
7 FPOL X+20 Y+30

8 FL IX+10 Y+20 RR F100

9 FCT PR+15 IPA+30 DR+ R15



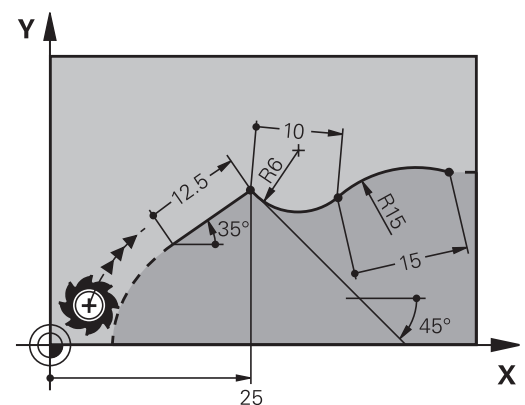
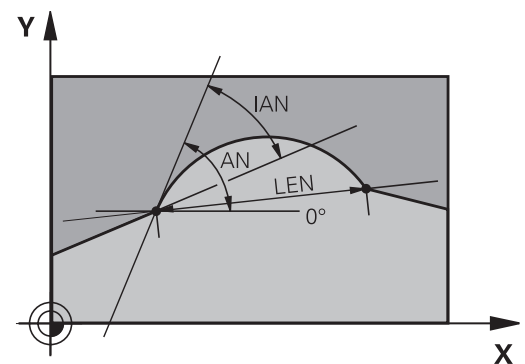
Direction and length of contour elements

Soft keys	Known data
	Length of a straight line
	Gradient angle of a straight line
	Chord length LEN of an arc
	Gradient angle AN of an entry tangent
	Center angle of an arc



Caution: Danger to the workpiece and tool!

Gradient angles that you defined incrementally (**IAN**) are referenced to the direction of the last positioning block by the TNC. Programs that contain incremental gradient angles and were created on an iTNC 530 or on earlier TNCs are not compatible.



Example NC blocks

27 FLT X+25 LEN 12.5 AN+35 RL F200

28 FC DR+ R6 LEN 10 AN-45

29 FCT DR- R15 LEN 15

Programming: Programming contours

6.6 Path contours – FK free contour programming

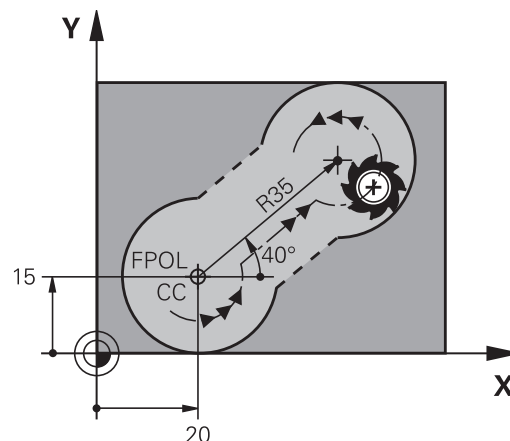
Circle center CC, radius and direction of rotation in the FC/FCT block

The TNC calculates a circle center for free-programmed arcs from the data you enter. This makes it possible to program full circles in an FK program block.

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 control encounters a block in which another **FPOL** is defined.

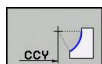
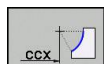


A circle center that was calculated or programmed conventionally is then no longer valid as a pole or circle center for the new FK contour: If you enter conventional polar coordinates that refer to a pole from a CC block you have defined previously, then you must enter the pole again in a CC block after the FK contour.

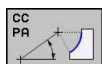
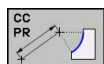


Soft keys

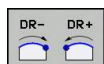
Known data



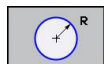
Circle center in Cartesian coordinates



Center point in polar coordinates



Rotational direction of the arc



Radius of an arc

Example NC blocks

```
10 FC CCX+20 CCY+15 DR+ R15
```

```
11 FPOL X+20 Y+15
```

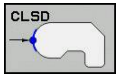
```
12 FL AN+40
```

```
13 FC DR+ R15 CCPR+35 CCPA+40
```


Closed contours

You can identify the beginning and end of a closed contour with the **CLSD** soft key. This reduces the number of possible solutions for the last contour element.

Enter **CLSD** as an addition to another contour data entry in the first and last blocks of an FK section.



Beginning of contour: CLSD+
End of contour: CLSD-

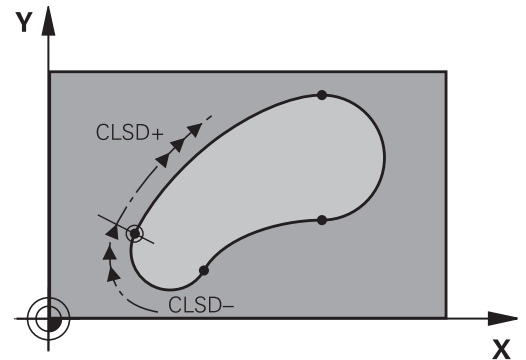
Example NC blocks

```
12 L X+5 Y+35 RL F500 M3
```

```
13 FC DR- R15 CLSD+ CCX+20 CCY+35
```

```
...
```

```
17 FCT DR- R+15 CLSD-
```



Programming: Programming contours

6.6 Path contours – FK free contour programming

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.

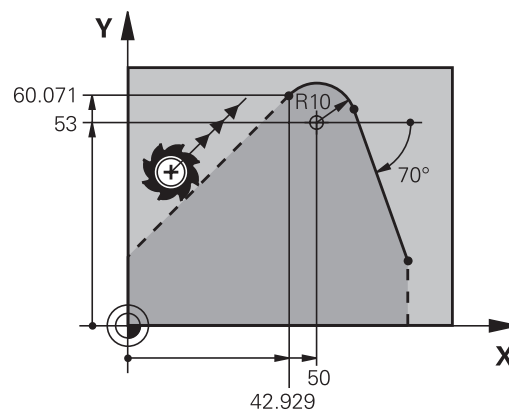
Auxiliary points on a contour

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 NC blocks

```
13 FC DR- R10 P1X+42.929 P1Y+60.071
```

```
14 FLT AN-70 PDX+50 PDY+53 D10
```


Relative data

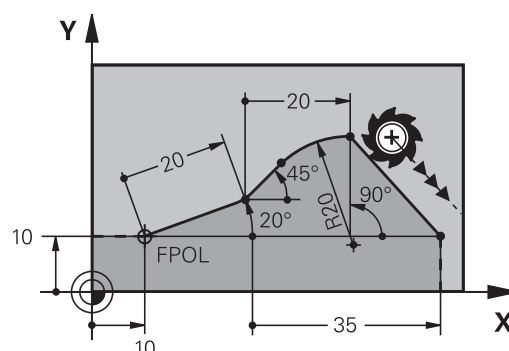
Data whose values are based on another contour element are called relative data. The soft keys and program words for entries begin with the letter **R** for **R**elative. The figure at right shows the entries that should be programmed as relative data.



The coordinates and angles for relative data are always programmed in incremental dimensions. You must also enter the 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 block in which you program the reference.

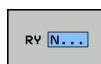
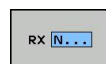
If you delete a block on which relative data are based, the TNC will display an error message. Change the program first before you delete the block.



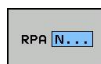
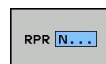
Data relative to block N: End point coordinates

Soft keys

Known data



Cartesian coordinates relative to block N



Polar coordinates relative to block N

Example NC blocks

12 FPOL X+10 Y+10

13 FL PR+20 PA+20

14 FL AN+45




15 FCT IX+20 DR- R20 CCA+90 RX 13

16 FL IPR+35 PA+0 RPR 13

Programming: Programming contours

6.6 Path contours – FK free contour programming

Data relative to block N: Direction and distance of the contour element

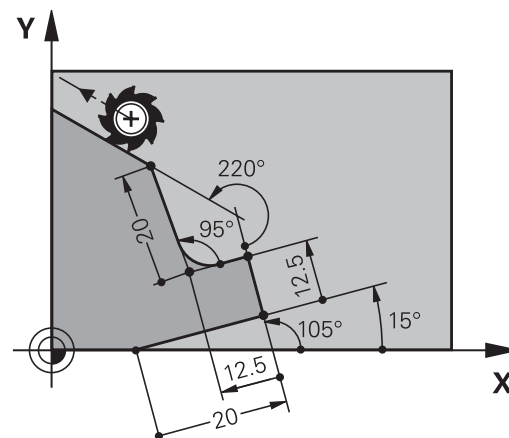
Soft key	Known data
 RAN [N...]	Angle between a straight line and another element or between the entry tangent of the arc and another element
 PAR [N...]	Straight line parallel to another contour element
 DP	Distance from a straight line to a parallel contour element

Example NC blocks





```

17 FL LEN 20 AN+15
18 FL AN+105 LEN 12.5
19 FL PAR 17 DP 12.5
20 FSELECT 2
21 FL LEN 20 IAN+95
22 FL IAN+220 RAN 18

```



Data relative to block N: Circle center CC

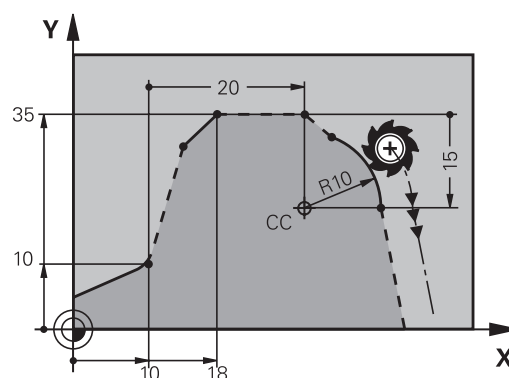
Soft key	Known data
 RCCX [N...]	Cartesian coordinates of the circle center relative to block N
 RCCY [N...]	
 RCCPR [N...]	Polar coordinates of the circle center relative to block N
 RCCPA [N...]	

Example NC blocks

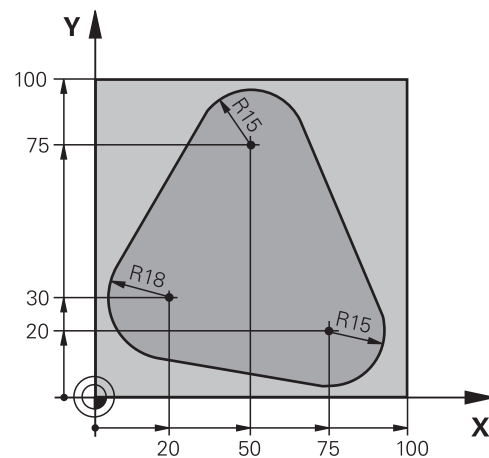
```

12 FL X+10 Y+10 RL
13 FL ...
14 FL X+18 Y+35
15 FL ...
16 FL ...
17 FC DR- R10 CCA+0 ICCX+20 ICCY-15 RCCX12 RCCY14

```



Example: FK programming 1

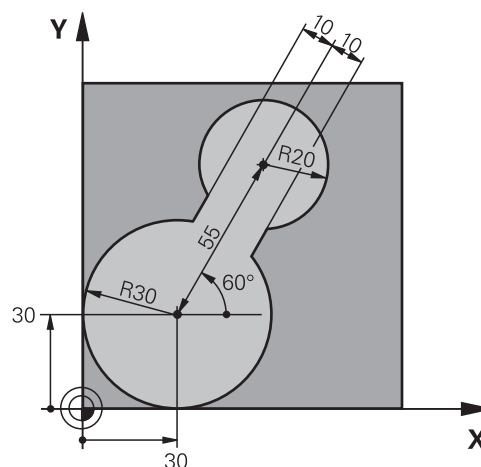


0 BEGIN PGM FK1 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	Definition of workpiece blank
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 1 Z S500	Tool call
4 L Z+250 R0 FMAX	Retract the tool
5 L X-20 Y+30 R0 FMAX	Pre-position the tool
6 L Z-10 R0 F1000 M3	Move to working depth
7 APPR CT X+2 Y+30 CCA90 R+5 RL F250	Approach the contour on a circular arc with tangential connection
8 FC DR- R18 CLSD+ CCX+20 CCY+30	FK contour section:
9 FLT	Program all known data for each contour element
10 FCT DR- R15 CCX+50 CCY+75	
11 FLT	
12 FCT DR- R15 CCX+75 CCY+20	
13 FLT	
14 FCT DR- R18 CLSD- CCX+20 CCY+30	
15 DEP CT CCA90 R+5 F1000	Depart the contour on a circular arc with tangential connection
16 L X-30 Y+0 R0 FMAX	
17 L Z+250 R0 FMAX M2	Retract the tool, end program
18 END PGM FK1 MM	

Programming: Programming contours

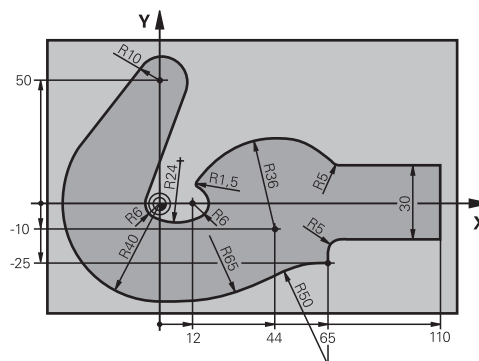
6.6 Path contours – FK free contour programming

Example: FK programming 2



0 BEGIN PGM FK2 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	Definition of workpiece blank
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 1 Z S4000	Tool call
4 L Z+250 R0 FMAX	Retract the tool
5 L X+30 Y+30 R0 FMAX	Pre-position the tool
6 L Z+5 R0 FMAX M3	Pre-position the tool
7 L Z-5 R0 F100	Move to working depth
8 APPR LCT X+0 Y+30 R5 RR F350	Approach the contour on a circular arc with tangential connection
9 FPOL X+30 Y+30	FK contour section:
10 FC DR- R30 CCX+30 CCY+30	Program all known data for each contour element
11 FL AN+60 PDX+30 PDY+30 D10	
12 FSELECT 3	
13 FC DR- R20 CCPR+55 CCPA+60	
14 FSELECT 2	
15 FL AN-120 PDX+30 PDY+30 D10	
16 FSELECT 3	
17 FC X+0 DR- R30 CCX+30 CCY+30	
18 FSELECT 2	
19 DEP LCT X+30 Y+30 R5	Depart the contour on a circular arc with tangential connection
20 L Z+250 R0 FMAX M2	Retract the tool, end program
21 END PGM FK2 MM	

Example: FK programming 3



0 BEGIN PGM FK3 MM	
1 BLK FORM 0.1 Z X-45 Y-45 Z-20	Definition of workpiece blank
2 BLK FORM 0.2 X+120 Y+70 Z+0	
3 TOOL CALL 1 Z S4500	Tool call
4 L Z+250 R0 FMAX	Retract the tool
5 L X-70 Y+0 R0 FMAX	Pre-position the tool
6 L Z-5 R0 F1000 M3	Move to working depth
7 APPR CT X-40 Y+0 CCA90 R+5 RL F250	Approach the contour on a circular arc with tangential connection
8 FC DR- R40 CCX+0 CCY+0	FK contour section:
9 FLT	Program all known data for each contour element
10 FCT DR- R10 CCX+0 CCY+50	
11 FLT	
12 FCT DR+ R6 CCX+0 CCY+0	
13 FCT DR+ R24	
14 FCT DR+ R6 CCX+12 CCY+0	
15 FSELECT 2	
16 FCT DR- R1.5	
17 FCT DR- R36 CCX+44 CCY-10	
18 FSELECT 2	
19 FCT DR+ R5	
20 FLT X+110 Y+15 AN+0	
21 FL AN-90	
22 FL X+65 AN+180 PAR21 DP30	
23 RND R5	
24 FL X+65 Y-25 AN-90	
25 FC DR+ R50 CCX+65 CCY-75	
26 FCT DR- R65	
27 FSELECT 1	
28 FCT Y+0 DR- R40 CCX+0 CCY+0	
29 FSELECT 4	
30 DEP CT CCA90 R+5 F1000	Depart the contour on a circular arc with tangential connection

6 Programming: Programming contours

6.6 Path contours – FK free contour programming

31 L X-70 R0 FMAX	
32 L Z+250 R0 FMAX M2	Retract the tool, end program
33 END PGM FK3 MM	

7

**Programming:
Data Transfer from
CAD Files**

Programming: Data Transfer from CAD Files

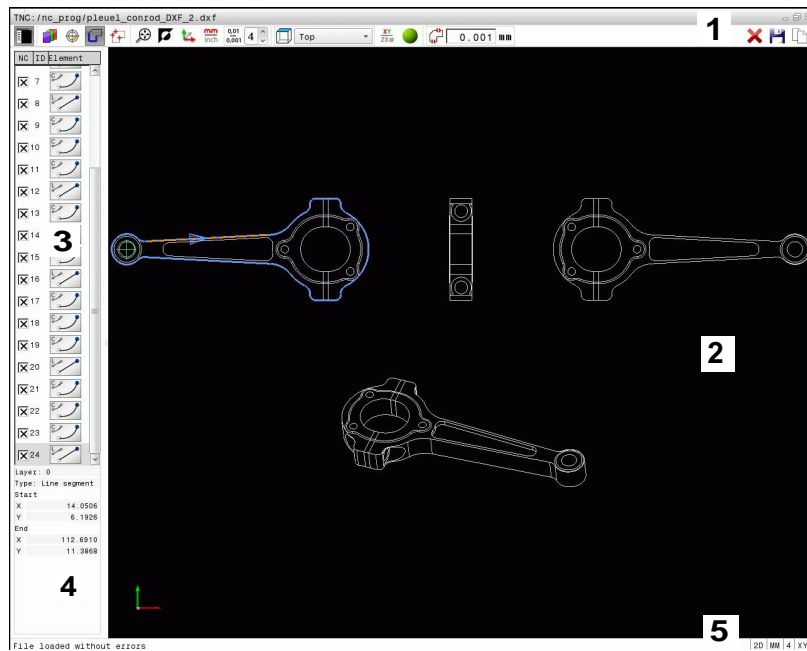
7.1 CAD viewer and DXF converter screen layout

7.1 CAD viewer and DXF converter screen layout

CAD viewer and DXF converter screen layout

If you open the CAD viewer or DXF converter, the following screen layout is displayed:

Screen display



- 1 Header
- 2 Graphics window
- 3 List view window
- 4 Element information window
- 5 Footer

7.2 CAD Viewer

Application

The CAD viewer allows you to open standardized CAD data formats directly on the TNC.











The TNC displays the following file formats:

Files	Type
Step files	.STP and .STEP
Iges files	.IGS and .IGES
DXF files	.DXF

The file can simply be selected via the file manager of the TNC, just like NC programs. This permits you to check quickly and simply for problems directly in the model.

You can position the datum anywhere in the model. In this way the coordinates of selected points can be displayed.

The following icons are available:

Icon	Setting
	Show or hide the list view window to enlarge the graphics window
	Display of the various layers
	Set the datum or delete set datum
	
	Set the zoom to the largest possible view of the complete graphics
	Change the background color (black or white)
	Set resolution: The resolution specifies how many decimal places the TNC should use when generating the contour program. Default setting: 4 decimal places with mm and 5 decimal places with inch
	Switch between various perspectives in the drawing e.g. Top
	Activate wire model selection or activate shadowing
	

Programming: Data Transfer from CAD Files

7.3 DXF converter (Option #42)

7.3 DXF converter (Option #42)

Application

DXF files can be opened directly by the TNC, in order to extract contours or machining positions, and save them as conversational programs or as point files. Conversational programs acquired in this manner can also be run by older TNC controls, since these contour programs contain only **L** and **CC/C** blocks.

If you process DXF files in the **Programming** operating mode, the TNC generates contour programs with the file extension **.H** and point files with the extension **.PNT** by default. However, you can choose the desired file type in the saving dialog. Furthermore, you can also save the selected contour or the selected machining positions to the clipboard of the TNC and then insert them directly in an NC program.

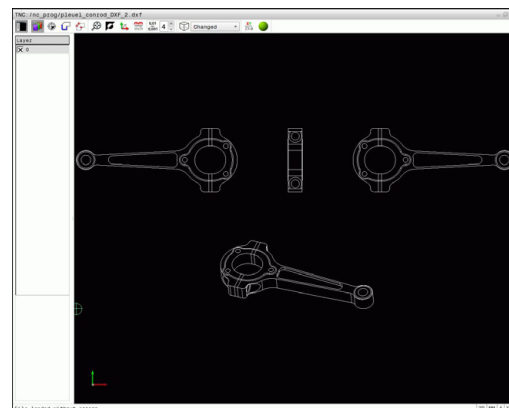


The file to be processed must be stored on the hard disk of your TNC.

Before loading the file to the TNC, ensure that the name of the file does not contain any blank spaces or illegal special characters, see "File names", page 115.

The TNC supports the most common DXF format, R12 (equivalent to AC1009).

The TNC does not support binary DXF format. When generating the DXF file from a CAD or drawing program, make sure that you save the file in ASCII format.



Working with the DXF converter



You cannot use the DXF converter without a mouse or touch pad. All operating modes and functions as well as contours and machining positions can only be selected with the mouse or touch pad.

The DXF converter runs as a separate application on the third desktop of the TNC. This enables you to use the screen switchover key to switch between the machine operating modes, the programming modes and the DXF converter as desired. This is especially useful if you want to insert contours or machining positions in a plain-language program by copying through the clipboard.

Opening a DXF file



- ▶ Select the **Programming** mode of operation



- ▶ Select File functions.



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



- ▶ In order to show all CAD files, press the **SHOW CAD** soft key

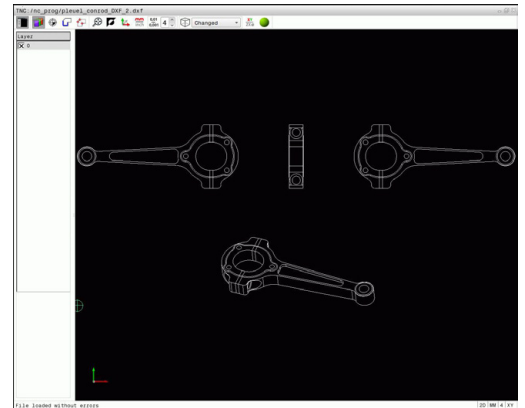


- ▶ Select the directory in which the CAD file is saved
- ▶ Select the desired CAD file
- ▶ Load it with the **ENT** key. The TNC starts the DXF converter and shows the contents of the file on the screen. The TNC shows the layers in the list view window, and the drawing in the graphics window

Basic settings

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




Icon	Setting
	Show or hide the list view window to enlarge the graphics window
	Display of the various layers
	Select the contour
	Select hole positions
	Set datum
	Set the zoom to the largest possible view of the complete graphics
	Change the background color (black or white)
	Switch between 2-D and 3-D mode. The active mode is color-highlighted
	Set the unit of measure (mm or inches) of the file. The TNC then outputs the contour program and the machining positions in this unit of measure. The active unit of measure is highlighted in red
	Set resolution: The resolution specifies how many decimal places the TNC should use when generating the contour program. Default setting: 4 decimal places with mm and 5 decimal places with inch
	Switch between various perspectives in the drawing e.g. Top
	Select a contour for a turning operation. The active machining is color-highlighted (option 50)
	Activate 3-D drawing wire model



Programming: Data Transfer from CAD Files

7.3 DXF converter (Option #42)

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

Icon	Setting
	<p>Contour assumption mode:</p> <p>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 0.0001 mm</p>
	<p>Point assumption mode:</p> <p>Specify whether the TNC should display the tool path as a dashed straight line during selection of machining positions</p>
	<p>Path optimization mode:</p> <p>The TNC optimizes the tool traverse movement to give the shortest traverse movements between the machining positions. Optimization is reset with repeated actuations</p>



Please note that you must set the correct unit of measure, since the DXF file does not contain any such information.

If you want to generate programs for older TNC controls, you must limit the resolution to three decimal places. In addition, you must remove the comments that the DXF converter inserts into the contour program.

The TNC displays the active basic settings in the footer of the screen.

Setting layers

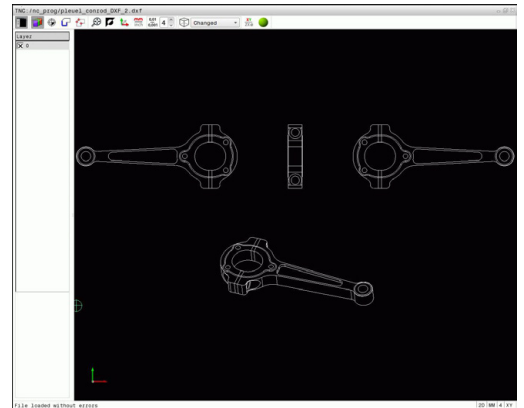
As a rule, DXF files contain multiple layers. The designer uses the layers to create groups of various types of elements, such as the actual workpiece contour, dimensions, auxiliary and design lines, shadings, and texts.

So that as little unnecessary information as possible appears on the screen during selection of the contours, you can hide all excessive layers contained in the DXF file.



The DXF file to be processed must contain at least one layer. Elements not assigned to a layer are automatically moved by the TNC to the "anonymous" layer.

You can even select a contour if the designer has saved it on different layers.



- ▶ Select the mode for the layer settings: In the list view window the TNC shows all layers contained in the active DXF 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

Programming: Data Transfer from CAD Files

7.3 DXF converter (Option #42)

Defining the datum

The datum of the drawing for the DXF file is not always located in a manner that lets you use it directly as a reference point for the workpiece. Therefore, the TNC has a function with which you can shift the drawing datum to a suitable location by clicking an element.

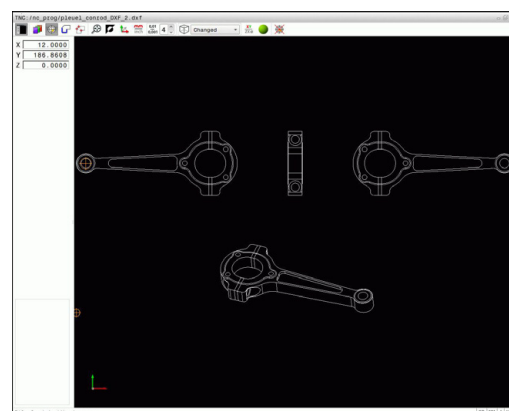
You can define a reference point at the following locations:

- 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
- By directly inputting numerical values into the list view window
- 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)



You must use the touchpad or a connected mouse in order to specify a reference point.

You can also change the reference point once you have already selected the contour. The TNC does not calculate the actual contour data until you save the selected contour in a contour program.



Selecting a reference point on a single element

- ▶ Select the mode for specifying the reference point
- ▶ Click the desired element with the mouse: The TNC indicates possible locations for reference points on the selected element with stars
- ▶ Click the star you want to select as reference point: The TNC sets the datum symbol at the selected place. Use the zoom function if the selected element is too small.

Selecting a reference point on the intersection of two elements


- ▶ Select the mode for specifying the reference point
- ▶ Click the first element (straight line, complete circle or circular arc) with the left mouse button. The TNC indicates possible locations for reference points on the selected element with stars. The element is color-highlighted
- ▶ Click the second element (straight line, complete circle or circular arc) with the left mouse button. The TNC sets the reference-point symbol on the intersection



The TNC calculates the intersection of two elements even if it is on the extension of one of these elements.

If the TNC calculates multiple intersections, it selects the intersection nearest the mouse-click on the second element.

If the TNC cannot calculate an intersection, it rescinds the marking of the element.

If a datum is set, the color of the icon  Set datum changes.

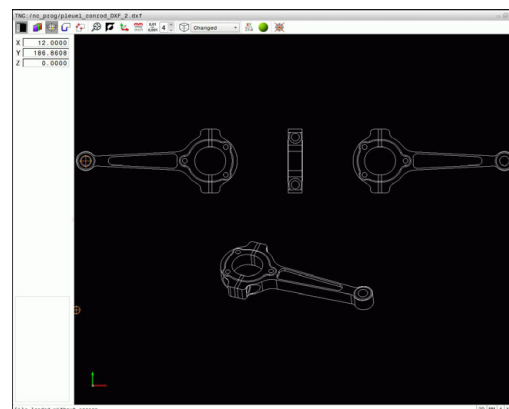
Delete a datum by clicking the icon .

Programming: Data Transfer from CAD Files

7.3 DXF converter (Option #42)

Element information

The TNC shows how far the reference point you haven chosen is located from the drawing datum.



Selecting and saving a contour

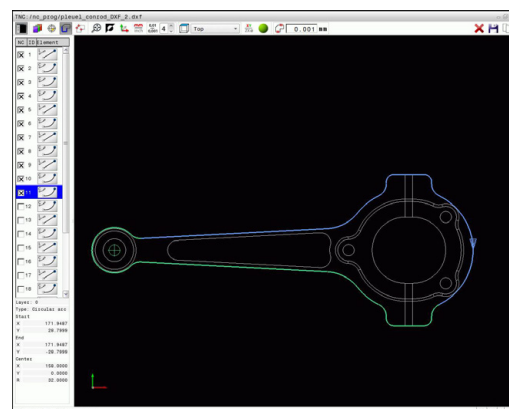


You must use the touchpad on the TNC keyboard or a mouse attached via the USB port in order to select a contour.

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 DXF elements are selectable as contours:

- LINE (straight line)
- CIRCLE (complete circle)
- ARC (circular arc)
- POLYLINE

Ellipses and splines can be used for intersections but cannot be selected. If you select ellipses or splines, these are displayed in red.

Element information

In the element information window, the TNC displays information about the contour element that you last selected via mouse click in the list view window or 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 mode for choosing a contour: The TNC hides the layers shown in the list view window. The graphics window is active for the contour selection
- ▶ To select a contour element: Click the desired element with the mouse. The TNC displays the machining sequence as a dashed straight 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, these elements turn green
- ▶ If further contour elements in the selected machining sequence are selectable, these elements turn green. With divergences, the element with the lowest angle distance is selected. Click on the last green element to assume all elements into the contour program
- ▶ The TNC shows all selected contour elements in the list view window. The TNC displays elements that are still green in the **NC** column without a check mark. The TNC does not save these elements to the contour program. You can also include the marked elements in the contour program by clicking 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. You can deselect all selected elements by clicking the icon



- ▶ Save the selected contour elements to the clipboard of the TNC so that you can then insert the contour in a plain-language program, or



- ▶ To save the selected contour elements in a plain-language program, enter any file name and the target directory in the pop-up window displayed by the TNC. Default setting: Name of the DXF file. As an alternative, you can also select the file type: Plain-language program (**.H**) or contour description (**.HC**)



- ▶ Confirm the entry: The TNC 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

Programming: Data Transfer from CAD Files

7.3 DXF converter (Option #42)



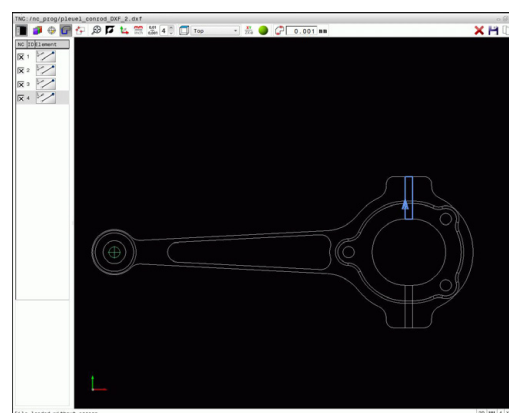
The TNC also transfers two workpiece-blank definitions (**BLK FORM**) to the contour program. The first definition contains the dimensions of the entire DXF file. The second one, which is the active one, contains only the selected contour elements, so that an optimized size of the workpiece blank results. The TNC only saves elements that have actually been selected (blue elements), which means that they have been given a check mark in the left 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 (with the shift key). A red star is shown as the starting point
- ▶ To select the next contour element: Click the desired element with the mouse. The TNC displays the machining sequence as a dashed straight line. When the element is selected the TNC displays it in blue. If the elements cannot be connected, the TNC displays the selected element in gray
- ▶ If further contour elements in the selected machining sequence are selectable, these elements turn green. With divergences, the element with the lowest angle distance is selected. Click on the last green element to assume all elements into the contour program



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 TNC extends/shortens the contour element along the same line. If the contour element to be extended or shortened is a circular arc, then the TNC extends/shortens the contour element along the same arc.

Select a contour for a turning operation

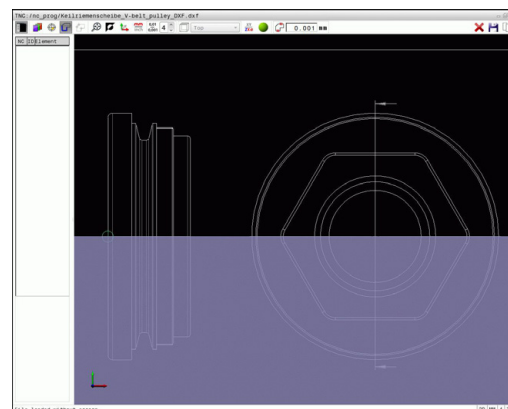
You can also use the DXF converter (option #50) to select contours for turning. The icon is dimmed if option #50 is not enabled. Before you enter a turning contour, you must set the datum onto the rotary axis. If you select a turning contour, it is saved with Z and X coordinates. In addition, all X coordinate 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 are not selected and are dimmed.



- ▶ Select the mode for choosing a turning contour: The TNC shows only the selectable elements above the rotation center
- ▶ Select the desired contour elements with the left mouse button: The TNC 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 dimmed.



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 TNC 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.

Programming: Data Transfer from CAD Files

7.3 DXF converter (Option #42)

Selecting and saving machining positions



You must use the touchpad on the TNC keyboard or a mouse attached via the USB port in order to select a machining position.

If the positions to be selected are very close to one another, use the zoom function.

If required, configure the basic settings so that the TNC shows the tool paths, see "Basic settings", page 263.

Three possibilities are available in the pattern generator for defining machining positions:

- Single selection: You select the desired machining position through individual mouse clicks (see "Single selection", page 273)
- 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 it (see "Rapid selection of hole positions with the mouse area", page 274).
- Quick selection of hole positions via an icon: Actuate the icon and the TNC then displays all existing hole diameters (see "Rapid selection of hole positions via icon", page 275).

Select the file type

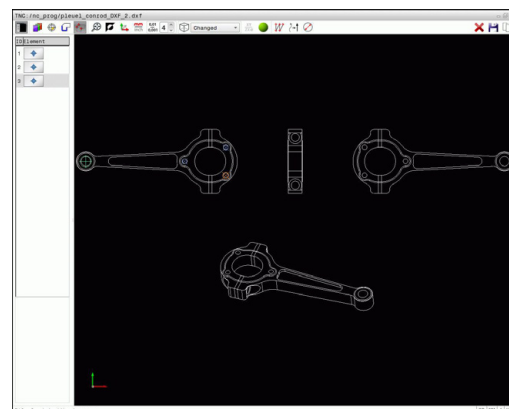
The following file types are available:

- Point table (.PNT)
- Plain-language program (.H)

If you save the machining positions to a plain-language program, the TNC creates a separate linear block with cycle call for every machining position (**L X... Y... M99**). You can also transfer this program to old TNC controls and run it there.



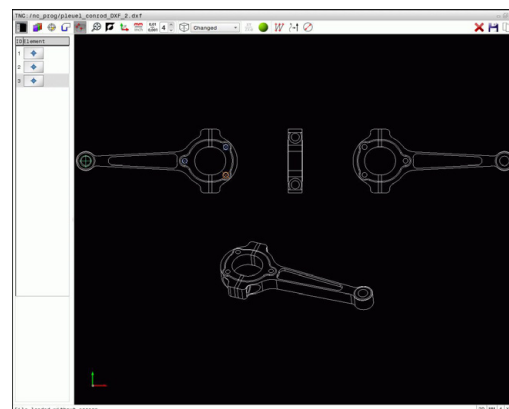
The point table (.PTN) from the TNC 640 is not compatible with the iTNC 530. Running the point table causes problems and unpredictable behavior.



Single selection



- ▶ Select the mode for choosing a machining position. The graphics window becomes active for position selection
- ▶ To select a machining position: Click the desired element with the mouse and the TNC displays the element in orange. If you simultaneously press the shift key, the TNC indicates possible machining positions on the element with stars. If you click a circle, the TNC adopts the circle center as machining position. If you simultaneously press the shift key, the TNC indicates possible machining positions with stars. The TNC loads the selected position into the list view window (and 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. Alternatively, select the element in the list view window and press **DEL**. You can deselect all selected elements by clicking the icon
- ▶ If you want to specify the machining position at the intersection of two elements, click the first element with the left mouse button: the TNC displays stars at the selectable machining positions.
- ▶ Click the second element (straight line, complete circle or circular arc) with the left mouse button. The TNC loads the intersection of the elements into the list view window (displays a point symbol). If there are several intersections, the TNC assumes the intersection nearest to the mouse.



- ▶ Save the selected machining positions to the clipboard of the TNC so that you can then insert them as a positioning block with cycle call in a plain-language program, or



- ▶ To save the selected machining positions to a point file, enter the target directory and any file name in the pop-up window displayed by the TNC. Default setting: Name of the DXF file. Alternately, you can also select the file type



- ▶ Confirm the entry: The TNC 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

Programming: Data Transfer from CAD Files

7.3 DXF converter (Option #42)

Rapid selection of hole positions with the mouse area



- ▶ Select the mode for choosing a machining position. The graphics window becomes active for position selection
- ▶ To select machining positions, press the shift key and define an area with the left mouse button. The TNC assumes all complete circles that are completely within the area as hole positions: The TNC opens a window in which you can filter the holes by size
- ▶ Configure the filter settings (see "Filter settings", page 276) and click the **OK** button to confirm: The TNC 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. Alternatively, select the element in the list view window and press **DEL**. If necessary you can also deselect elements that you already selected, by dragging an area open again, but this time while pressing the **CTRL** key



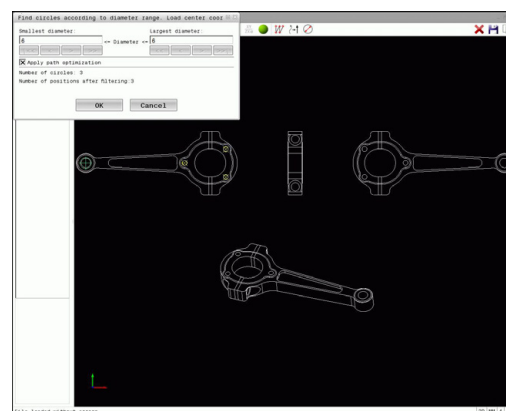
- ▶ Save the selected machining positions to the clipboard of the TNC so that you can then insert them as a positioning block with cycle call in a plain-language program, or
- ▶ To save the selected machining positions to a point file, enter the target directory and any file name in the pop-up window displayed by the TNC. Default setting: Name of the DXF file. Alternately, you can also select the file type



- ▶ Confirm the entry: The TNC 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 a machining position. The graphics window becomes active for position selection



- ▶ Select the icon: The TNC opens a window in which you can filter the holes by size
- ▶ If required, configure the filter settings (see "Filter settings", page 276) and click the **OK** button to confirm: The TNC 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. Alternatively, select the element in the list view window and press **DEL**. You can deselect all selected elements by clicking the icon



- ▶ Save the selected machining positions to the clipboard of the TNC so that you can then insert them as a positioning block with cycle call in a plain-language program, or



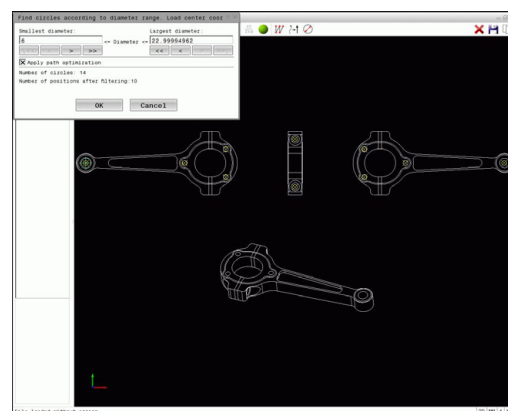
- ▶ To save the selected machining positions to a point file, enter the target directory and any file name in the pop-up window displayed by the TNC. Default setting: Name of the CAD file. Alternately, you can also select the file type



- ▶ Confirm the entry: The TNC 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



Programming: Data Transfer from CAD Files

7.3 DXF converter (Option #42)

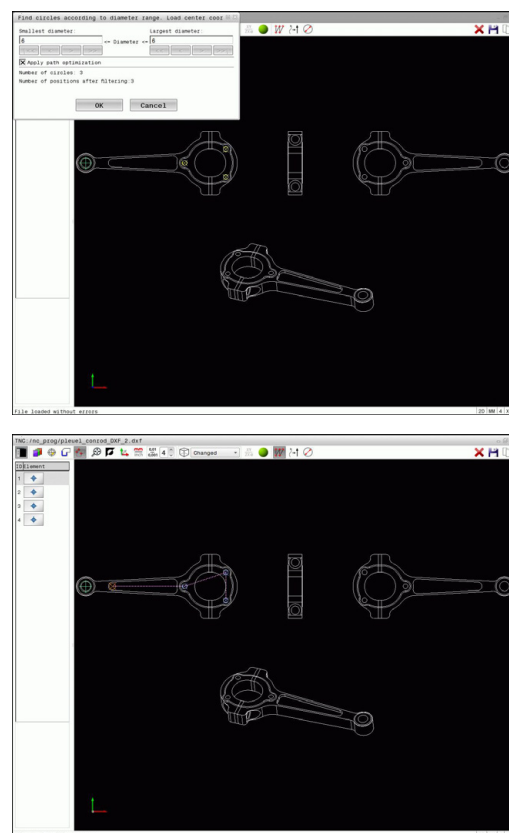
Filter settings

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 TNC 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 TNC 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 path displayed by clicking the **Show tool path** icon, see "Basic settings", page 263.

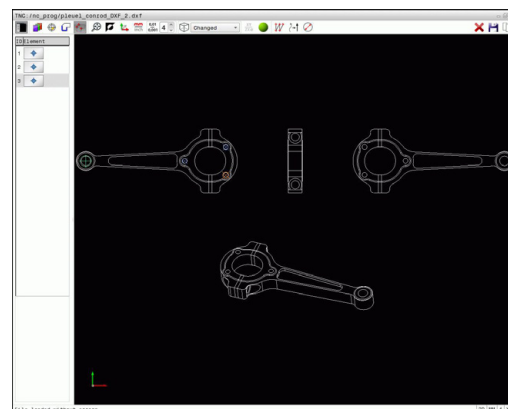


Element information

In the element information window, the TNC displays the coordinates of the machining position that you last selected via mouse click in the list view window or graphics window.

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

- ▶ In order to rotate the model shown in three dimensions you hold the right mouse button down and move the mouse.
- ▶ 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 TNC zooms in on the defined area.
- ▶ To rapidly magnify or reduce any area: Rotate the mouse wheel backwards or forwards.
- ▶ To return to the standard display: Press the shift key and simultaneously double-click with the right mouse key. The rotation angle is maintained if you only double-click with the right mouse key.



8

**Programming:
Subprograms and
program section
repeats**

Programming: Subprograms and program section repeats

8.1 Labeling 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 are marked in a part program by labels (**LBL**).

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 program with the **LABEL SET** key. The number of label names you can enter is only limited by the internal memory.



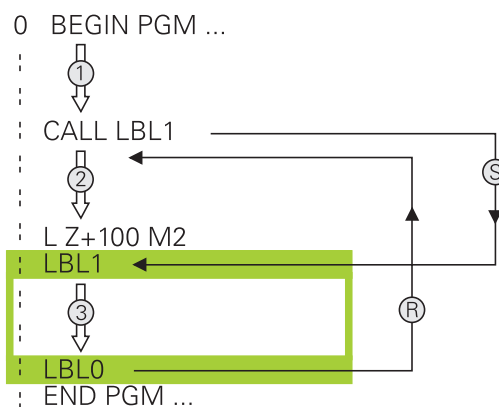
Do not use a label number or label name more than once!

Label 0 (**LBL 0**) 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 TNC executes the part program up to calling a subprogram, **CALL LBL**.
- 2 The subprogram is then executed from beginning to end, **LBL 0**.
- 3 The TNC then resumes the part program from the block after the subprogram call **CALL LBL**.



Programming notes

- A main program can contain any number of subprograms
- You can call subprograms in any sequence and as often as desired
- A subprogram cannot call itself
- Write subprograms after the block with M2 or M30
- If subprograms are located before the block with M2 or M30 in the part program, they will be executed at least once even if they are not called

Programming a 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
- ▶ To mark the end, press the LBL SET key and enter the label number "0"

8.2 Subprograms

Calling a subprogram

LBL
CALL

- ▶ Call a subprogram: Press the **LBL CALL** key
- ▶ Enter the subprogram number of the subprogram you wish to call. If you want to use a label name, press the **LBL NAME** soft key to switch to text entry.
- ▶ If you want to enter the number of a string parameter as target address, press the QS soft key. The TNC will then jump to the label name that is specified in the defined string parameter
- ▶ Ignore repeats **REP** by pressing the **NO ENT** key. Repeat **REP** is used only for program section repeats

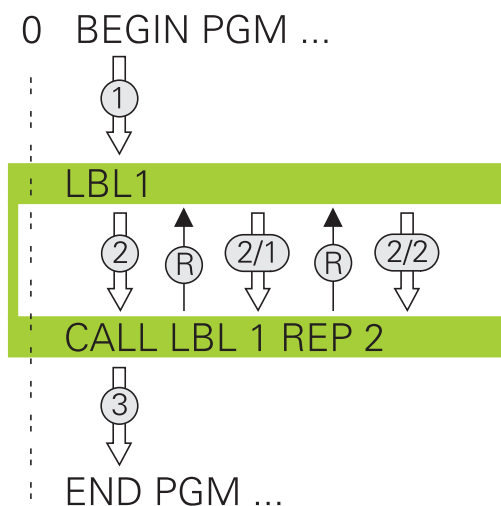


CALL LBL 0 is not permitted as it is only used to call the end of a subprogram.

8.3 Program-section repeats

Label

The beginning of a program section repeat is marked by the label **LBL**. The end of a program section repeat is identified by **CALL LBL n REPn**.



Operating sequence

- 1 The TNC executes the part program up to the end of the program section (**CALL LBL n REPn**)
- 2 Then the program section between the called LABEL and the label call **CALL LBL n REPn** is repeated the number of times entered after **REP**
- 3 The TNC resumes the part 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



- ▶ 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

Programming: Subprograms and program section repeats

8.3 Program-section repeats

Calling a program section repeat

LBL
CALL

- ▶ Call a program section: Press the LBL CALL key
- ▶ Enter the 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 program as subprogram

Overview of the soft keys

If the **PGM CALL** key is pressed, the TNC displays the following soft keys:

Soft key	Function
CALL PROGRAM	Call a program with PGM CALL
SELECT DATUM TABLE	Select a datum table with SEL TABLE
SELECT POINT TABLE	Select a point table with SEL PATTERN
SELECT CONTOUR	Select a contour program with SEL CONTOUR
SELECT PROGRAM	Select a program with SEL PGM
CALL SELECTED PROGRAM	Select last selected file with CALL SELECTED PGM

Programming: Subprograms and program section repeats

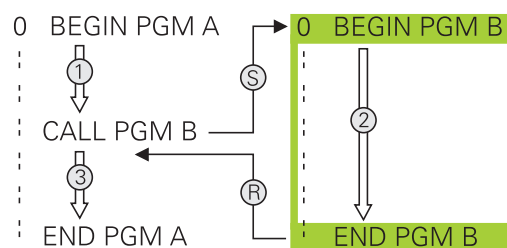
8.4 Any desired program as subprogram

Operating sequence

- 1 The TNC executes the part program up to the block in which another program is called with **CALL PGM**
- 2 Then the other part program is run from beginning to end
- 3 The TNC then resumes the first part program (i.e. the calling program) with the block after the program call



If you want to program variable program calls in connection with string parameters, use the **SEL PGM** function.



Programming notes

- The TNC does not need any labels to call any part program
- The called program must not contain the miscellaneous functions **M2** or **M30**. If you have defined subprograms with labels in the called part program, you then need to replace M2 or M30 with the **FN 9: IF +0 EQU +0 GOTO LBL 99** jump function to force a jump over this program section
- The called part program must not contain a **CALL PGM** call into the calling part program, otherwise an infinite loop will result

Calling any program as a subprogram



Danger of collision!

Coordinate transformations that you define in the called program remain in effect for the calling program too, unless you reset them.



If the program you want to call is located in the same directory as the program you are calling it from, then you only need to enter the program name.

If the called program is not located in the same directory as the program you are calling it from, you must enter the complete path, e.g. **TNC: \ZW35\ROUGH\PGM1.H**

If you want to call a DIN/ISO program, enter the file type **.I** after the program name.

You can also call a program with Cycle **12 PGM CALL**.

As a rule, Q parameters are effective globally with a **PGM CALL**. So please note that changes to Q parameters in the called program also influence the calling program.

Call a program with PGM CALL

The **PGM CALL** function calls any program as a subprogram. The control runs the called program from the position where it was called in the program.

PGM
CALL

- To select the functions for program call, press the **PGM CALL** key

CALL
PROGRAM

- Press the **CALL PROGRAM** soft key for the TNC to start the dialog for defining the program to be called. Enter the path name with the keyboard, or

SELECT
FILE

- press the **SELECT FILE** soft key for the TNC to display a selection window in which you can select the program to be called. Confirm with the **END** key

Programming: Subprograms and program section repeats




8.4 Any desired program as subprogram

Call with **SEL PGM** and **CALL SELECTED PGM**



Use the function **SEL PGM** to select any program as a subprogram and call it at another position in the program. The control runs the called program from the position where it was called in the program with **CALL SELECTED PGM**.

The **SEL PGM** function is also permitted with string parameters, so that you can dynamically control program calls.

To select the program, proceed as follows:

- 
 - ▶ To select the functions for program call, press the **PGM CALL** key
- 
 - ▶ Press the **SELECT PROGRAM** soft key for the TNC to start the dialog for defining the program to be called.
- 
 - ▶ press the **SELECT FILE** soft key for the TNC to display a selection window in which you can select the program to be called. Confirm with the **END** key

To call the selected program, proceed as follows:

- 
 - ▶ To select the functions for program call, press the **PGM CALL** key
- 
 - ▶ Press the **CALL SELECTED PROGRAM** soft key for the TNC to call the previously selected program with **CALL SELECTED PGM**

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 **CYCL CALL** acts like a main program call
- You can nest program section repeats as often as desired

Programming: Subprograms and program section repeats

8.5 Nesting

Subprogram within a subprogram

Example NC blocks

0 BEGIN PGM UPGMS MM	
...	
17 CALL LBL "SP1"	Call the subprogram marked with LBL SP1
...	
35 L Z+100 R0 FMAX M2	Last program block of the main program with M2
36 LBL "SP1"	Beginning of subprogram SP1
...	
39 CALL LBL 2	Call the subprogram marked with LBL 2
...	
45 LBL 0	End of subprogram 1
46 LBL 2	Beginning of subprogram 2
...	
62 LBL 0	End of subprogram 2
63 END PGM UPGMS MM	

Program execution

- 1 Main program UPGMS is executed up to block 17.
- 2 Subprogram SP1 is called, and executed up to block 39.
- 3 Subprogram 2 is called, and executed up to block 62. End of subprogram 2 and return jump to the subprogram from which it was called.
- 4 Subprogram 1 is called, and executed from block 40 up to block 45. End of subprogram 1 and return jump to the main program UPGMS.
- 5 Main program UPGMS is executed from block 18 up to block 35. Return jump to block 1 and end of program.

Repeating program section repeats

Example NC blocks

0 BEGIN PGM REPS MM	
...	
15 LBL 1	Beginning of program section repeat 1
...	
20 LBL 2	Beginning of program section repeat 2
...	
27 CALL LBL 2 REP 2	Program section call with two repeats
...	
35 CALL LBL 1 REP 1	The program section between this block and LBL 1
...	(block 15) is repeated once
50 END PGM REPS MM	

Program execution

- 1 Main program REPS is executed up to block 27.
- 2 Program section between block 27 and block 20 is repeated twice.
- 3 Main program REPS is executed from block 28 to block 35.
- 4 Program section between block 35 and block 15 is repeated once (including the program section repeat between 20 and block 27).
- 5 Main program REPS is executed from block 36 to block 50.
Return jump to block 1 and end of program

Programming: Subprograms and program section repeats

8.5 Nesting

Repeating a subprogram

Example NC blocks

0 BEGIN PGM UPGREP MM	
...	
10 LBL 1	Beginning of program section repeat 1
11 CALL LBL 2	Subprogram call
12 CALL LBL 1 REP 2	Program section call with two repeats
...	
19 L Z+100 R0 FMAX M2	Last block of the main program with M2
20 LBL 2	Beginning of subprogram
...	
28 LBL 0	End of subprogram
29 END PGM UPGREP MM	

Program execution

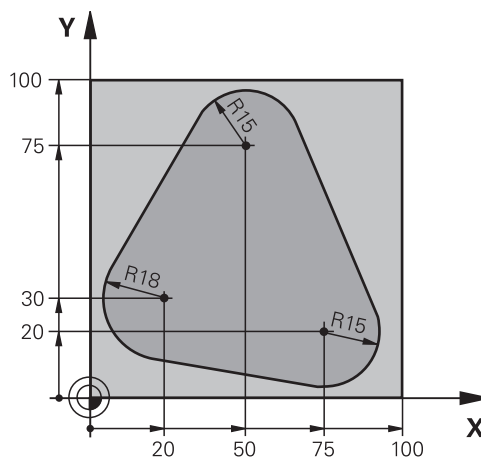
- 1 Main program UPGREP is executed up to block 11.
- 2 Subprogram 2 is called and executed.
- 3 Program section between block 12 and block 10 is repeated twice. This means that subprogram 2 is repeated twice.
- 4 Main program UPGREP is executed from block 13 up to block 19. Return jump to block 1 and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

8.6 Programming examples

Example: Milling a contour in several infeeds

Program sequence:

- Pre-position the tool to the workpiece surface
- Enter the infeed depth in incremental values
- Contour milling
- Repeat infeed and contour-milling



0 BEGIN PGM PGMWDH MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-40	
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 1 Z S500	Tool call
4 L Z+250 R0 FMAX	Retract the tool
5 L X-20 Y+30 R0 FMAX	Pre-position in the working plane
6 L Z+0 R0 FMAX M3	Pre-position to the workpiece surface
7 LBL 1	Set label for program section repeat
8 L IZ-4 R0 FMAX	Infeed depth in incremental values (in space)
9 APPR CT X+2 Y+30 CCA90 R+5 RL F250	Contour approach
10 FC DR- R18 CLSD+ CCX+20 CCY+30	Contour
11 FLT	
12 FCT DR- R15 CCX+50 CCY+75	
13 FLT	
14 FCT DR- R15 CCX+75 CCY+20	
15 FLT	
16 FCT DR- R18 CLSD- CCX+20 CCY+30	
17 DEP CT CCA90 R+5 F1000	Contour departure
18 L X-20 Y+0 R0 FMAX	Retract tool
19 CALL LBL 1 REP 4	Return jump to LBL 1; section is repeated a total of 4 times
20 L Z+250 R0 FMAX M2	Retract the tool, end program
21 END PGM PGMWDH MM	

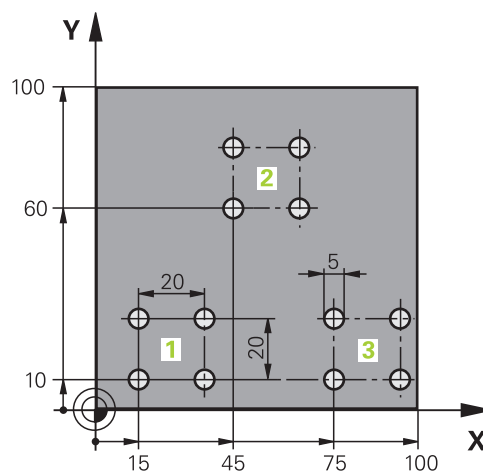
Programming: Subprograms and program section repeats

8.6 Programming examples

Example: Groups of holes

Program sequence:

- Approach the groups of holes in the main program
- Call the group of holes (subprogram 1) in the main program
- Program the group of holes only once in subprogram 1

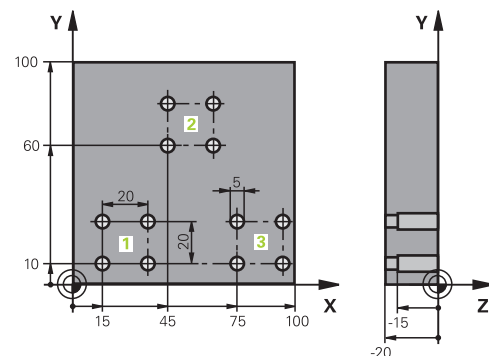


0 BEGIN PGM UP1 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 1 Z S5000	Tool call
4 L Z+250 R0 FMAX	Retract the tool
5 CYCL DEF 200	Cycle definition: drilling
Q200=2 ;SET-UP CLEARANCE	
Q201=-10 ;DEPTH	
Q206=250 ;FEED RATE FOR PLNGNG	
Q202=5 ;	
Q210=0 ;DWELL TIME AT TOP	
Q203=+0 ;SURFACE COORDINATE	
Q204=10 ;2ND SET-UP CLEARANCE	
Q211=0.25 ;DWELL TIME AT DEPTH	
6 L X+15 Y+10 R0 FMAX M3	Move to starting point for group 1
7 CALL LBL 1	Call the subprogram for the group
8 L X+45 Y+60 R0 FMAX	Move to starting point for group 2
9 CALL LBL 1	Call the subprogram for the group
10 L X+75 Y+10 R0 FMAX	Move to starting point for group 3
11 CALL LBL 1	Call the subprogram for the group
12 L Z+250 R0 FMAX M2	End of main program
13 LBL 1	Beginning of subprogram 1: Group of holes
14 CYCL CALL	Hole 1
15 L IX+20 R0 FMAX M99	Move to 2nd hole, call cycle
16 L IY+20 R0 FMAX M99	Move to 3rd hole, call cycle
17 L IX-20 R0 FMAX M99	Move to 4th hole, call cycle
18 LBL 0	End of subprogram 1
19 END PGM UP1 MM	

Example: Group of holes with several tools

Program sequence:

- Program the fixed cycles in the main program
- Call the complete hole pattern (subprogram 1) in the main program
- Approach the groups of holes (subprogram 2) in subprogram 1
- Program the group of holes only once in subprogram 2



0 BEGIN PGM SP2 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 1 Z S5000	Centering drill tool call
4 L Z+250 R0 FMAX	Retract the tool
5 CYCL DEF 200	Cycle definition: CENTERING
Q200=2 ;SET-UP CLEARANCE	
Q202=-3 ;DEPTH	
Q206=250 ;FEED RATE FOR PLNGNG.	
Q202=3 ;	
Q210=0 ;DWELL TIME AT TOP	
Q203=+0 ;SURFACE COORDINATE	
Q204=10 ;2ND SET-UP CLEARANCE	
Q211=0.25 ;DWELL TIME AT DEPTH	
6 CALL LBL 1	Call subprogram 1 for the entire hole pattern
7 L Z+250 R0 FMAX	
8 TOOL CALL 2 Z S4000	Drill tool call
9 FN 0: Q201 = -25	New depth for drilling
10 FN 0: Q202 = +5	New plunging depth for drilling
11 CALL LBL 1	Call subprogram 1 for the entire hole pattern
12 L Z+250 R0 FMAX	
13 TOOL CALL 3 Z S500	Reamer tool call

Programming: Subprograms and program section repeats

8.6 Programming examples

14 CYCL DEF 201 REAMING	Cycle definition: REAMING
Q200=2 ;SET-UP CLEARANCE	
Q201=-15 ;DEPTH	
Q206=250 ;FEED RATE FOR PLNGNG.	
Q211=0.5 ;DWELL TIME AT DEPTH	
Q208=400 ;RETRACTION FEED RATE	
Q203=+0 ;SURFACE COORDINATE	
Q204=10 ;2ND SET-UP CLEARANCE	
15 CALL LBL 1	Call subprogram 1 for the entire hole pattern
16 L Z+250 R0 FMAX M2	End of main program
17 LBL 1	Beginning of subprogram 1: Entire hole pattern
18 L X+15 Y+10 R0 FMAX M3	Move to starting point for hole group 1
19 CALL LBL 2	Call subprogram 2 for the hole group
20 L X+45 Y+60 R0 FMAX	Move to starting point for hole group 2
21 CALL LBL 2	Call subprogram 2 for the hole group
22 L X+75 Y+10 R0 FMAX	Move to starting point for hole group 3
23 CALL LBL 2	Call subprogram 2 for the hole group
24 LBL 0	End of subprogram 1
25 LBL 2	Beginning of subprogram 2: Group of holes
26 CYCL CALL	1st hole with active fixed cycle
27 L IX+20 R0 FMAX M99	Move to 2nd hole, call cycle
28 L IY+20 R0 FMAX M99	Move to 3rd hole, call cycle
29 L IX-20 R0 FMAX M99	Move to 4th hole, call cycle
30 LBL 0	End of subprogram 2
31 END PGM SP2 MM	

9

**Programming: Q
Parameters**

9.1 Principle and overview of functions

9.1 Principle and overview of functions

With parameters you can program entire families of parts in a single part program, by programming variable parameters instead of fixed numerical values.

Use parameters for e.g.:

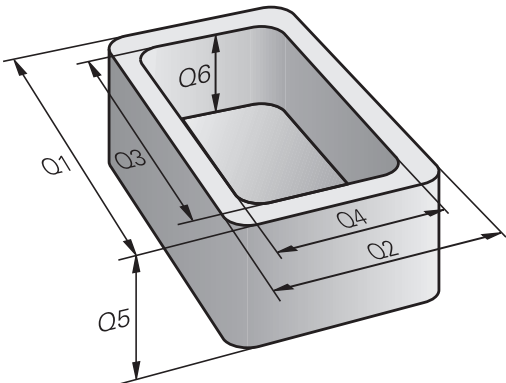
- Coordinate values
- Feed rates
- Spindle speeds
- Cycle data

With parameters you can also:

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

Parameters are always identified with letters and numbers. The letters determine the type of parameter and the numbers the parameter range.

See the table below for detailed information:



Parameter type	Parameter range	Meaning
Q parameters:		Parameters effect all programs in the TNC memory
	0 - 30	Parameters for HEIDENHAIN cycles
	31 - 99	Parameters for users
	100 - 199	Parameters for special TNC functions
	200 - 1199	Parameters for HEIDENHAIN cycles
	1200 - 1399	Parameters for cycles of machine tool builder or third party provider
	1400 - 1499	Parameters for CALL-active cycles of machine tool builder or third party provider
	1500 - 1599	Parameters for DEF-active cycles of machine tool builder or third party provider
	1600 - 1999	Parameters for users
QL parameters		Parameters only effective locally within a program
	0 - 499	Parameters for users
QR parameters		Parameters that are nonvolatile on all programs in the TNC memory, i.e. they remain in effect even after a power interruption
	0 - 499	Parameters for users

QS parameters (the **S** stands for string) are also available on the TNC and enable you to process texts.

Parameter type	Parameter range	Meaning
QS parameters	Parameters effect all programs in the TNC memory	
	0 - 99	Parameters for users
	100 - 199	Parameters for system information on the TNC that can be read by the NC programs of the user or by cycles
	200 - 1199	Parameters for HEIDENHAIN cycles
	1200 - 1399	Parameters providing feedback to the user's NC program with cycles of the machine tool builder or a third party provider
	1400 - 1599	Parameters for cycles of machine tool builder or third party provider
	1600 - 1999	Parameters for users



You gain maximum safety for your applications by using only parameter ranges recommended for the user in your NC programs.

Please note that the specified use of the parameter ranges is recommended by HEIDENHAIN but cannot be ensured.

Machine tool builder or third party functions may still cause overlaps with the user's NC program. In this regard, please observe the machine manual or third-party documentation.

9.1 Principle and overview of functions

Programming notes

You can mix Q parameters and fixed numerical values within a 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 TNC calculates numbers up to a value of 10^{10} .

You can assign a maximum of 255 characters to **QS** parameters.

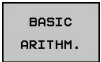
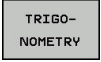
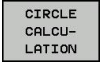






The TNC always assigns some Q and QS parameters the same data. For example the Q parameter **Q108** is always assigned the current tool radius, see "Preassigned Q parameters", page 356.

The TNC saves numerical values internally in a binary number format (standard IEEE 754). Due to this standardized format some decimal numbers do not have an exact binary representation (round-off error). Keep this in mind especially when you use calculated Q-parameter contents for jump commands or positioning movements.

Calling Q parameter functions

When you are writing a part program, press the "Q" key (in the numeric keypad for numerical input and axis selection, below the +/-). The TNC then displays the following soft keys:

Soft key	Function group	Page
	Basic arithmetic (assign, add, subtract, multiply, divide, square root)	303
	Trigonometric functions	305
	Function for calculating circles	306
	If/then conditions, jumps	307
	Other functions	311
	Entering formulas in the part program	341
	Function for machining complex contours	See User's Manual for Cycles



The TNC shows the soft keys Q, QL and QR when you are defining or assigning a Q parameter. First press one of these soft keys to select the desired type of parameter, and then enter the parameter number.

If you have a USB keyboard connected, you can press the Q key to open the dialog for entering a formula.

9.2

Part families—Q parameters in place of numerical values

9.2

Part families—Q parameters in place of numerical values

Application

The Q parameter function **FN 0: ASSIGN** assigns numerical values to Q parameters. This enables you to use variables in the program instead of fixed numerical values.

Example NC blocks

15 FN 0: Q10=25	Assign
...	Q10 is assigned the value 25
25 L X +Q10	Means L X +25

You need write only one program for a whole family of parts, entering the characteristic dimensions as Q parameters.

To program a particular part, you then assign the appropriate values to the individual Q parameters.

Example: Cylinder with Q parameters

- Cylinder radius:

R = Q1
- Cylinder height:

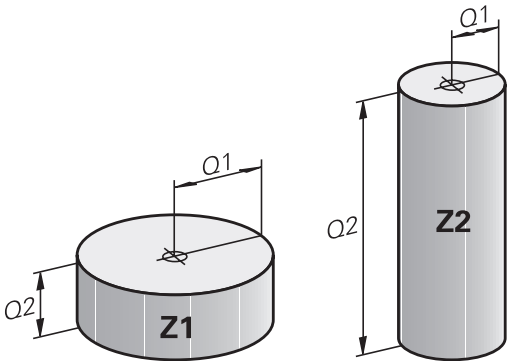
H = Q2
- Cylinder Z1:

Q1 = +30

Q2 = +10
- Cylinder Z2:

Q1 = +10

Q2 = +50



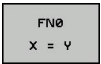
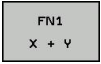
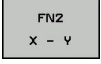

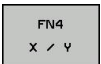

9.3 Describing contours with mathematical functions

Application

The Q parameters listed below enable you to program basic mathematical functions in a part program:

- Select a Q-parameter function: Press the Q key (in the numerical keypad at right). The Q-parameter functions are displayed in a soft-key row
- Select the mathematical functions: Press the **BASIC ARITHMETIC** soft key. The TNC then displays the following soft keys:

Overview

Soft key	Function
	FN 0: ASSIGN e.g. FN 0: Q5 = +60 Directly assign value
	FN 1: ADDITION e.g. FN 1: Q1 = -Q2 + -5 Form and assign sum from two values
	FN 2: SUBTRACTION e.g. FN 2: Q1 = +10 - +5 Form and assign difference between two values
	FN 3: MULTIPLICATION e.g. FN 3: Q2 = +3 * +3 Form and assign the product of two values
	FN 4: DIVISION e.g. FN 4: Q4 = +8 DIV +Q2 Form and assign the quotient of two values Not permitted: Division by 0
	FN 5: SQUARE ROOT e.g. FN 5: Q20 = SQRT 4 Form and assign the square root of a value Not permitted: Square root from negative value

To the right of the "=" character you can enter the following:

- 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.

9.3

Describing contours with mathematical functions

Programming fundamental operations

Example 1

Q

▶ Select the Q parameter functions: Press the Q key

BASIC
ARITHM.

▶ Select the mathematical functions: Press the BASIC ARITHMETIC soft key

FN0
X = Y

▶ Select the Q parameter function ASSIGN: Press the FN0 X = Y soft key

Program blocks in the TNC

16 FN 0: Q5 = +10
17 FN 3: Q12 = +Q5 * +7

PARAMETER NUMBER FOR RESULT?

ENT

▶ **12** Enter the Q parameter number and confirm with the **ENT** key

FIRST VALUE / PARAMETER?

ENT

▶ Enter **10**: Assign the numerical value 10 to Q5 and confirm with the **ENT** soft key.

Example 2

Q

▶ Select the Q parameter functions: Press the Q key

BASIC
ARITHM.

▶ Select the mathematical functions: Press the BASIC ARITHMETIC soft key

FN3
X * Y

▶ To select the Q parameter function MULTIPLICATION, press the FN3 X * Y soft key

PARAMETER NUMBER FOR RESULT?

ENT

▶ **12** Enter the Q parameter number 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.

9.4 Angle functions

Definitions

Sine: $\sin \alpha = a / c$

Cosine: $\cos \alpha = b / c$

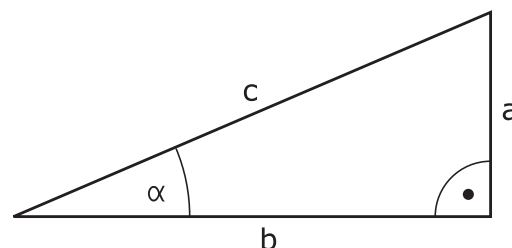
Tangent: $\tan \alpha = a / b = \sin \alpha / \cos \alpha$

where

- c is the side opposite the right angle
- a is the side opposite the angle α
- b is the third side.

The TNC can find the angle from the tangent:

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



Example:

$a = 25 \text{ mm}$

$b = 50 \text{ mm}$

$$\alpha = \arctan (a / b) = \arctan 0.5 = 26.57^\circ$$

Furthermore:

$$a^2 + b^2 = c^2 \text{ (where } a^2 = a \times a \text{)}$$

$$c = \sqrt{a^2 + b^2}$$

Programming trigonometric functions

Press the soft key to call the trigonometric functions. The TNC then displays the soft keys that are listed in the table below.

Soft key	Function
<div style="border: 1px solid black; padding: 2px; text-align: center;"> FN6 SIN(X) </div>	FN 6: SINUS e. g. FN 6: Q20 = SIN-Q5 Define and assign the sinus of an angle in degrees (°)
<div style="border: 1px solid black; padding: 2px; text-align: center;"> D7 COS(X) </div>	FN 7: COSINUS e. g. FN 7: Q21 = COS-Q5 Define and assign the cosine of an angle in degrees (°)
<div style="border: 1px solid black; padding: 2px; text-align: center;"> FN8 X LEN Y </div>	FN 8: ROOT SUM OF SQUARES e. g. FN 8: Q10 = +5 LEN +4 Form and assign length from two values
<div style="border: 1px solid black; padding: 2px; text-align: center;"> FN13 X ANG Y </div>	FN 13: ANGLE e. g. FN 13: Q20 = +25 ANG-Q1 Calculate the angle from the arc tangent of two sides or from the sine and cosine of the angle ($0 < \text{angle} < 360^\circ$) and assign it to a parameter

9.5

Calculation of circles

9.5

Calculation of circles

Application

The TNC 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 if you wish to determine the location and size of a hole or a pitch circle using the programmable probing function.

Soft key	Function
<div>FN23 3 POINTS OF CIRCLE</div>	FN 23: Determining the CIRCLE DATA from three points e. g. FN 23: Q20 = CDATA Q30


The coordinate pairs of three points on a circle must be saved in Q30 and the following five parameters—in this case, up to Q35.

The TNC 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
<div>FN24 4 POINTS OF CIRCLE</div>	FN 24: Determining the CIRCLE DATA from four points e. g. FN 24: Q20 = CDATA Q30

The coordinate pairs of four points on a circle must be saved in Q30 and the following seven parameters—in this case, up to Q37.

The TNC then saves the circle center in the reference axis (X if spindle axis is Z) in parameter Q20, the circle center in the minor axis (Y if spindle axis is Z) in parameter Q21, and the circle radius in parameter Q22.



Note that **FN 23** and **FN 24** automatically overwrite the resulting parameter and the two following parameters.

9.6 If-then decisions with Q parameters

Application

The TNC 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 TNC continues the program at the label that is programmed after the condition (for information on labels, see "Labeling Subprograms and Program Section Repeats", page 280). If it is not fulfilled, the TNC continues with the next block.

To call another program as a subprogram, enter a **PGM CALL** program call after the block with the target label.

Unconditional jumps

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

FN 9: IF+10 EQU+10 GOTO LBL1

Abbreviations used:

IF	:	If
EQU	:	Equal to
NE	:	Not equal to
GT	:	Greater than
LT	:	Less than
GOTO	:	Go to
UNDEFINED	:	Undefined
DEFINED	:	Defined

Programming: Q Parameters

9.6 If-then decisions with Q parameters

Programming if-then decisions

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

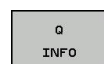
Soft key	Function
<div>FN9 IF X EQ Y GOTO</div>	FN 9: IF EQUAL, JUMP e. g. FN 9: IF +Q1 EQU +Q3 GOTO LBL "UPCAN25" If both values or parameters are equal, jump to specified label
<div>EQU</div>	
<div>FN9 IF X EQ Y GOTO</div>	FN 9: IF UNDEFINED, JUMP e. g. FN 9: IF +Q1 IS UNDEFINED GOTO LBL "UPCAN25" If the given parameter is undefined, jump to the specified label
<div>IS UNDEFINED</div>	
<div>FN9 IF X EQ Y GOTO</div>	FN 9: IF DEFINED, JUMP e. g. FN 9: IF +Q1 IS DEFINED GOTO LBL "UPCAN25" If the given parameter is defined, jump to the specified label
<div>IS DEFINED</div>	
<div>FN10 IF X NE Y GOTO</div>	FN 10: IF UNEQUAL, JUMP e. g. FN 10: IF +10 NE -Q5 GOTO LBL 10 If both values or parameters are unequal, jump to specified label
<div>FN11 IF X GT Y GOTO</div>	FN 11: IF GREATER, JUMP g. g. FN 11: IF+Q1 GT+10 GOTO LBL 5 If the first value or parameter is greater than the second value or parameter, jump to specified label
<div>FN12 IF X LT Y GOTO</div>	FN 12: IF LESS, JUMP e. g. FN 12: IF+Q5 LT+0 GOTO LBL "ANYNAME" If the first value or parameter is smaller than the second value or parameter, jump to specified label

9.7 Checking and changing Q parameters

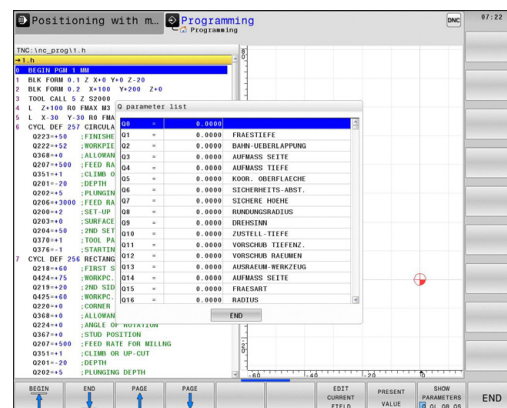
Procedure

You can check Q parameters in all operating modes, and also edit them.

- If you are in a program run, interrupt it if required (for example, by pressing the machine STOP button and the **soft** key). If you are in a test run, interrupt it.



- To call the Q parameter functions, press the **Q** **INFO** soft key or the **Q** key
- The TNC lists all parameters and their 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 **soft** key, enter the new value, and confirm with the **ENT** key.
- To leave the value unchanged, press the **soft** key or end the dialog with the **END** key.



The parameters used by the TNC internally or in cycles are provided with comments.

If you want to check or edit local, global or string parameters, press the **SHOW PARAMETERS Q QL QR QS** soft key. The TNC then displays the specific parameter type. The functions previously described also apply.

Programming: Q Parameters

9.7 Checking and changing Q parameters

You can have the Q parameters be shown in the additional status display in all operating modes (except for the **Programming** operating mode).

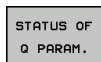
- ▶ If you are in a program run, interrupt it if required (for example, by pressing the machine STOP button and the soft key). If you are in a test run, interrupt it.



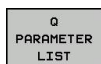
- ▶ Call the soft-key row for screen layout



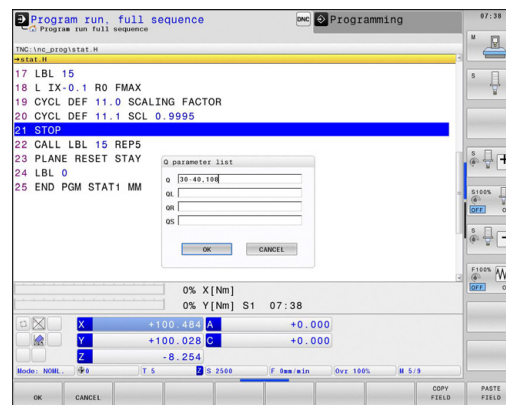
- ▶ Select the screen layout with additional status display: In the right half of the screen, the TNC shows the **Overview** status form



- ▶ Press the **STATUS OF Q PARAM.** soft key



- ▶ Press the **Q PARAMETER LIST** soft key: The TNC 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 TNC then displays the following soft keys:

Soft key	Function	Page
	FN 14: ERROR Display error messages	312
	FN 16: F-PRINT Formatted output of texts or Q parameter values	316
	FN 18: SYSREAD Read system data	320
	FN 19: PLC Transfer values to the PLC	329
	FN 20: WAIT FOR NC and PLC synchronization	329
	FN 29: PLC Transfer up to eight values to the PLC	330
	FN 37: EXPORT Export local Q parameters or QS parameters into a calling program	330
	FN 26: TABOPEN Open a freely definable table	430
	FN 27: TABWRITE Write to a freely definable table	431
	FN 28: TABREAD Read from a freely definable table	432

9.8 Additional functions

FN 14: ERROR: Displaying error messages

With the function **FN 14: ERROR** you can call messages under program control. The messages are predefined by the machine tool builder or by HEIDENHAIN. Whenever the TNC comes to a block with **FN 14: ERROR** in the Program Run or Test Run mode, it interrupts the program run and displays a message. The program must then be restarted. The error numbers are listed in the table.

Range of error numbers	Standard dialog text
0 ... 999	Machine-dependent dialog
1000 ... 1199	Internal error messages (see table)

Example NC block

The TNC is to display the text stored under error number 1000:

```
180 FN 14: ERROR = 1000
```

Error message predefined by HEIDENHAIN

Error number	Text
1000	Spindle?
1001	Tool axis is missing
1002	Tool radius too small
1003	Tool radius too large
1004	Range exceeded
1005	Start position incorrect
1006	ROTATION not permitted
1007	SCALING FACTOR not permitted
1008	MIRROR IMAGE not permitted
1009	Datum shift not permitted
1010	Feed rate is missing
1011	Input value incorrect
1012	Incorrect sign
1013	Entered angle not permitted
1014	Touch point inaccessible
1015	Too many points
1016	Contradictory input
1017	CYCL incomplete
1018	Plane wrongly defined
1019	Wrong axis programmed
1020	Wrong rpm
1021	Radius comp. undefined
1022	Rounding-off undefined
1023	Rounding radius too large
1024	Program start undefined


Error number	Text
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
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

9.8 Additional functions

Error number	Text
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 allowed
1094	Tool name not allowed
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

Error number	Text
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

FN16: F-PRINT – Formatted output of text and Q parameter values



With **FN16: F-PRINT**, you can also output to the screen any messages from the NC program. Such messages are displayed by the TNC in a pop-up window.

The function **FN16: F-PRINT** transfers Q parameter values and texts in a selectable format. If you send the values, the TNC saves the data in the file that you defined in the **FN16** block. The maximum size of the output file is 20 kilobytes.

To output the formatted texts and Q-parameter values, create a text file with the TNC’s text editor. In this file you then define the output format and Q parameters you want to output.

```

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 = %9.3LF", Q31;
"Y1 = %9.3LF", Q32;
"Z1 = %9.3LF", Q33;
    
```

When you create a text file, use the following formatting functions:

Special characters	Function
"....."	Define output format for texts and variables between the quotation marks
%9.3LF	Define the format for Q parameters: 9 total characters (incl. decimal point), of which 3 are after the decimal, Long, Floating (decimal number)
%S	Format for text variable
%d	Format for integer
,	Separation character between output format and parameter
;	End of block character
\n	Line break

The following functions allow you to include the following additional information in the protocol log file:

Keyword	Function
CALL_PATH	Indicates the path for the NC program where you will find the FN16 function. Example: "Measuring program: %S",CALL_PATH;
M_CLOSE	Closes the file to which you are writing with FN16. 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 text only for English conversational language
L_GERMAN	Outputs text only for German conversational 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_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_PORTUGUE	Outputs text only for Portuguese conversational language
L_HUNGARIA	Outputs text only for Hungarian conversational language
L_SLOVENIAN	Outputs text only for Slovenian conversational language
L_ALL	Display text independently of the conversational language

9.8 Additional functions

Keyword	Function
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

In the part program, program **FN 16: F-PRINT** to activate the output:

```
96 FN 16: F-PRINT TNC:\MASKE\MASKE1.A/ TNC:\PROT1.TXT
```

The TNC then creates the file PROT1.TXT:

MEASURING LOG OF IMPELLER CENTER OF GRAVITY

DATE: 27.09.2014

TIME: 8:56:34 AM

NO. OF MEASURED VALUES : = 1

X1 = 149.360

Y1 = 25.509

Z1 = 37.000



If you output the same file more than once in the program, the TNC appends all texts to the end of the texts already output within the target file.

If you use **FN16** more than once in the program, the TNC saves all texts in the file that you defined in the **FN16** function. The file is not output until the TNC reads the **END PGM** block, or you press the NC stop button, or you close the file with **M_CLOSE**.

In the **FN16** block, program the format file and the log file with their respective file type extensions

If you enter only the file name for the path of the log file, the TNC saves the log file in the directory in which the NC program with the **FN16** function is located.

You can define a standard path for outputting protocol files via the user parameters **fn16DefaultPath** and **fn16DefaultPathSim** (Program Test).

Displaying messages on the TNC screen

You can also use the function **FN16: F-PRINT** to display any messages from the NC program in a pop-up window on the TNC screen. This makes it easy to display explanatory texts, including long texts, at any point in the 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 TNC screen, you need only enter **SCREEN:** as the name of the protocol file.

96 FN 16: F-PRINT TNC:\MASKE\MASKE1.A\SCREEN:

If the message has more lines than fit in the pop-up window, you can use the arrow keys to page in the window.

To close the pop-up window, press the **CE** key. To have the program close the window, program the following NC block:

96 FN 16: F-PRINT TNC:\MASKE\MASKE1.A\SCLR:



If you output the same file more than once in the program, the TNC appends all texts to the end of the texts already output within the target file.

Exporting messages

The **FN 16** function also enables you to save the log files externally.

Enter the complete target path in the **FN 16** function:

96 FN 16: F-PRINT TNC:\MSK\MSK1.A / PC325:\LOG\PRO1.TXT



If you output the same file more than once in the program, the TNC appends all texts to the end of the texts already output within the target file.

9.8 Additional functions

FN 18: SYSREAD: Reading system data

With the **FN 18: SYSREAD** function you can read system data and store them in Q parameters. You select the system data through a group name (ID number), and additionally through a number and an index.

Group name, ID no.	Number	Index	Meaning
Program information, 10	3	-	Number of the active fixed cycle
	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.
System jump addresses, 13	1	-	Label jumped to during M2/M30 instead of ending the current program. Value = 0: M2/M30 has the normal effect
	2	-	Label jumped to if FN14: ERROR after the NC CANCEL reaction instead of aborting the program with an error. The error number programmed in the FN14 command can be read under ID992 NR14. Value = 0: FN14 has the normal effect.
	3	-	Label jumped to in the event of an internal server error (SQL, PLC, CFG) instead of aborting the program with an error message. Value = 0: Server error has the normal effect.
Machine status, 20	1	-	Active tool number
	2	-	Prepared tool number
	3	-	Active tool axis 0=X, 1=Y, 2=Z, 6=U, 7=V, 8=W
	4	-	Programmed spindle speed
	5	-	Active spindle condition: -1=not defined, 0=M3 active, 1=M4 active, 2=M5 after M3, 3=M5 after M4
	7	-	Gear range
	8	-	Coolant status: 0=off, 1=on
	9	-	Active feed rate
	10	-	Index of prepared tool
	11	-	Index of active tool
Channel data, 25	1	-	Channel number

Group name, ID no.	Number	Index	Meaning
Cycle parameter, 30	1	-	Set-up clearance of active fixed cycle
	2	-	Drilling depth / milling depth of active fixed cycle
	3	-	Plunging depth of active fixed cycle
	4	-	Feed rate for pecking in active fixed cycle
	5	-	1st side length for rectangular pocket cycle
	6	-	2nd side length for rectangular pocket cycle
	7	-	1st side length for slot cycle
	8	-	2nd side length for slot cycle
	9	-	Radius for circular pocket cycle
	10	-	Feed rate for milling in active fixed cycle
	11	-	Direction of rotation for active fixed cycle
	12	-	Dwell time for active fixed cycle
	13	-	Thread pitch for Cycles 17, 18
	14	-	Finishing allowance for active fixed cycle
	15	-	Direction angle for rough out in active fixed cycle
Modal condition, 35	21	-	Probing angle
	22	-	Probing path
	23	-	Probing feed rate
Data for SQL tables, 40	1	-	Dimensions: 0 = absolute (G90) 1 = incremental (G91)
Data from the tool table, 50	1	Tool no.	Tool length
	2	Tool no.	Tool radius
	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 (0 or 1)
	8	Tool no.	Number of the replacement tool

9.8 Additional functions

Group name, ID no.	Number	Index	Meaning
	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: Rotational direction DIRECT (0=positive/-1=negative)
	19	Tool no.	TT: Offset in plane R-OFFS
	20	Tool no.	TT: Offset in length L-OFFS
	21	Tool no.	TT: Break tolerance for length LBREAK
	22	Tool no.	TT: Break tolerance for radius RBREAK
	23	Tool no.	PLC value
	25	Tool no.	Probe center offset in minor axis (CAL-OF ₂)
	26	Tool no.	Spindle angle during calibration (CAL-ANG)
	27	Tool no.	Tool type for pocket table
	28	Tool no.	Maximum rpm NMAX
	32	Tool no.	Point angle TANGLE
	34	Tool no.	LIFTOFF allowed (0= No, 1= Yes)
	35	Tool no.	Wear tolerance for radius R2TOL
	37	Tool no.	Corresponding line in the touch-probe table
	38	Tool no.	Timestamp of last use
Pocket table data, 51	1	Pocket number	Tool number
	2	Pocket number	Special tool: 0=No, 1=Yes
	3	Pocket number	Fixed pocket: 0=No, 1=Yes
	4	Pocket number	Locked pocket: 0=No, 1=Yes
	5	Pocket number	PLC status
Tool pocket, 52	1	Tool no.	Pocket number P
	2	Tool no.	Magazine number

Group name, ID no.	Number	Index	Meaning
Values programmed immediately after TOOL CALL, 60	1	-	Tool number T
	2	-	Active tool axis 0 = X 6 = U 1 = Y 7 = V 2 = Z 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
Values programmed immediately after TOOL DEF, 61	1	-	Tool number T
	2	-	Length
	3	-	Radius
	4	-	Index
	5	-	Tool data programmed in TOOL DEF 1 = Yes, 0 = No
Active 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

9 Programming: Q Parameters

9.8 Additional functions

Group name, ID no.	Number	Index	Meaning
Active transformations, 210	1	-	Basic rotation in MANUAL OPERATION mode
	2	-	Programmed rotation with Cycle 10
	3	-	Active mirrored axis
			0: Mirroring not active
			+1: X axis mirrored
			+2: Y axis mirrored
			+4: Z axis mirrored
			+64: U axis mirrored
			+128: V axis mirrored
			+256: W axis mirrored
			Combinations = Sum of individual axes
	4	1	Active scaling factor in X axis
	4	2	Active scaling factor in Y axis
	4	3	Active scaling factor in Z axis
	4	7	Active scaling factor in U axis
	4	8	Active scaling factor in V axis
	4	9	Active scaling factor in W axis
	5	1	3-D ROT A axis
	5	2	3-D ROT B axis
	5	3	3-D ROT C axis
	6	-	Tilted working plane active / inactive (-1/0) in a Program Run operating mode
	7	-	Tilted working plane active / inactive (-1/0) in a Manual operating mode
Active datum shift, 220	2	1	X axis
		2	Y axis
		3	Z axis
		4	A axis
		5	B axis
		6	C axis
		7	U axis
		8	V axis
		9	W axis

Group name, ID no.	Number	Index	Meaning
Traverse range, 230	2	1 to 9	Negative software limit switch in axes 1 to 9
	3	1 to 9	Positive software limit switch in axes 1 to 9
	5	-	Software limit switch on or off: 0 = on, 1 = off
Nominal position in the REF system, 240	1	1	X axis
		2	Y axis
		3	Z axis
		4	A axis
		5	B axis
		6	C axis
		7	U axis
		8	V axis
		9	W axis
Current position in the active coordinate system, 270	1	1	X axis
		2	Y axis
		3	Z axis
		4	A axis
		5	B axis
		6	C axis
		7	U axis
		8	V axis
		9	W axis
Interpretation of coordinates in turning operation, 310	20	1 to 3	Coordinates are give with respect to: 0 = diameter, -1 = radius
		(X, Y, Z)	

9.8 Additional functions

Group name, ID no.	Number	Index	Meaning
TS triggering touch probe, 350	50	1	Touch probe type
		2	Line in the touch-probe table
	51	-	Effective length
	52	1	Effective ball radius
		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
	56	1	Maximum measuring range
		2	Safety clearance
	57	1	Spindle orientation possible: 0=No, 1=Yes
		2	Spindle-orientation angle
TT tool touch probe	70	1	Touch probe type
		2	Line in the touch-probe table
	71	1	Center point in reference axis (REF system)
		2	Center point in minor axis (REF system)
		3	Center point in tool axis (REF system)
	72	-	Probe contact radius
	75	1	Rapid traverse
		2	Measuring feed rate for stationary spindle
		3	Measuring feed rate for rotating spindle
	76	1	Maximum measuring range
		2	Safety clearance for linear measurement
		3	Safety clearance for radial measurement
	77	-	Spindle speed
	78	-	Probing direction

Group name, ID no.	Number	Index	Meaning
Reference point from touch probe cycle, 360	1	1 to 9 (X, Y, Z, A, B, C, U, V, W)	Last reference point of a manual touch probe cycle, or last touch point from Cycle 0 without probe length compensation but with probe radius compensation (workpiece coordinate system)
	2	1 to 9 (X, Y, Z, A, B, C, U, V, W)	Last reference point of a manual touch probe cycle, or last touch point from Cycle 0 without probe length or probe radius compensation (machine coordinate system)
	3	1 to 9 (X, Y, Z, A, B, C, U, V, W)	Result of measurement of the touch probe cycles 0 and 1 without probe radius or probe length compensation
	4	1 to 9 (X, Y, Z, A, B, C, U, V, W)	Last reference point of a manual touch probe cycle, or last touch point from Cycle 0 without probe length or stylus probe compensation (workpiece coordinate system)
	10	-	Oriented spindle stop
Value from the active datum table in the active coordinate system, 500	Line	Column	Read values
Basic transformation, 507	Line	1 to 6 (X, Y, Z, SPA, SPB, SPC)	Read the basic transformation of a preset
Axis offset, 508	Line	1 to 9 (X_OFFS, Y_OFFS, Z_OFFS, A_OFFS, B_OFFS, C_OFFS, U_OFFS, V_OFFS, W_OFFS)	Read the axis offset of a preset
Active preset, 530	1	-	Read the number of the active preset
Read data of the 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
	10	-	Maximum tool age TIME2
	11	-	Current tool age CUR. TIME

9.8 Additional functions

Group name, ID no.	Number	Index	Meaning
	12	-	PLC status
	13	-	Maximum tooth length 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
	20	-	TT: Offset in length L-OFFS
	21	-	TT: Break tolerance for length LBREAK
	22	-	TT: Break tolerance for radius RBREAK
	23	-	PLC value
	24	-	Tool type TYP 0 = milling cutter, 21 = touch probe
	27	-	Corresponding line in the touch-probe table
	32	-	Point angle
	34	-	Lift off
Touch probe cycles, 990	1	-	Approach behaviour: 0 = standard behavior 1 = effective radius, safety clearance zero
	2	-	0 = Pushbutton monitoring off 1 = Pushbutton monitoring on
	4	-	0 = Stylus not deflected 1 = Stylus deflected
	8	-	Current spindle angle
Execution status, 992	10	-	Mid-program startup active 1 = yes, 0 = no
	11	-	Search phase
	14	-	Number of the last FN14 error
	16	-	Real execution active 1 = execution , 2 = simulation
	31	-	Radius compensation in MDI mode with paraxial positioning blocks permitted 0 = not permitted, 1 = permitted

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

55 FN 18: SYSREAD Q25 = ID210 NR4 IDX3

FN 19: PLC – Transfer values to the PLC



This function may only be used with the permission of your machine tool builder.

The **FN 19: PLC** function transfers up to two numerical values or Q parameters to the PLC.

FN 20: WAIT FOR – NC and PLC synchronization



This function may only be used with the permission of your machine tool builder.

With the **FN 20: WAIT FOR** function you can synchronize the NC and PLC during a program run. The NC stops machining until the condition that you have programmed in the **FN 20: WAIT FOR** block is fulfilled.

SYNC is used whenever you read, for example, system data via **FN18: SYSREAD** that require synchronization with real time. The TNC stops the look-ahead calculation and executes the subsequent NC block only when the NC program has actually reached that block.

Example: Pause internal look-ahead calculation, read current position in the X axis

```
32 FN 20: WAIT FOR SYNC
```

```
33 FN 18: SYSREAD Q1 = ID270 NR1 IDX1
```


9.8 Additional functions

FN 29: PLC – Transfer values to the PLC



This function may only be used with the permission of your machine tool builder.

The **FN 29: PLC** function transfers up to eight numerical values or Q parameters to the PLC.

FN 37: EXPORT



This function may only be used with the permission of your machine tool builder.

You need the **FN 37: EXPORT** function if you want to create your own cycles and integrate them in the TNC.

9.9 Accessing tables with SQL commands

Introduction

Accessing of tables is programmed on the TNC with SQL commands in **transactions**. A transaction consists of multiple SQL commands that guarantee an orderly execution of the table entries.



Tables are configured by the machine manufacturer. Names and designations required as parameters for SQL commands are also specified.

The following **terms** are used:

- **Table:** A table consists of x columns and y rows. It is saved as a file in the File Manager of the TNC, and is addressed with the path and file name (=table name). Synonyms can also be used for addressing, as an alternative to the path and file name.
- **Columns:** The number and names of the columns are specified when configuring the table. In some SQL commands the column name is used for addressing.
- **Rows:** The number of rows is variable. You can insert new rows. There are no row numbers or other designators. However, you can select rows based on the contents of a column. Rows can only be deleted in the table editor, not by an NC program.
- **Cell:** The part of a column in a row.
- **Table entry:** Content of a cell.
- **Result set:** During a transaction, the selected columns and rows are managed in the result set. You can view the result set as a sort of "intermediate memory," which temporarily assumes the set of selected columns and rows. Result set
- **Synonym:** This term defines a name used for a table instead of its path and file name. Synonyms are specified by the machine manufacturer in the configuration data.

9.9 Accessing tables with SQL commands

A transaction

In principle, a transaction consists of the following actions:

- Address the table (file), select rows and transfer them to the result set.
- Read rows from the result set, change rows or insert new rows.
- Conclude transaction: If changes/insertions were made, the rows from the result set are placed in the table (file).

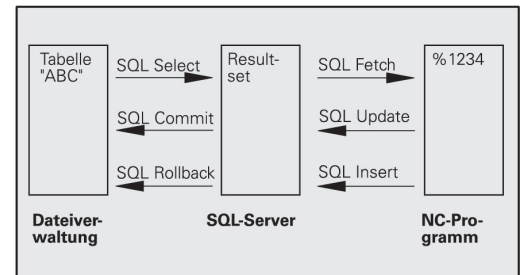
Other actions are also necessary so that table entries can be edited in an NC program and to ensure that other changes are not made to copies of the same table rows at the same time. This results in the following **transaction sequence**:

- 1 A Q parameter is specified for each column to be edited. The Q parameter is assigned to a column—it is "bound" (**SQL BIND...**
- 2 Address the table (file), select rows and transfer them to the result set. In addition, you define which columns are transferred to the result set (**SQL SELECT...**). You can lock the selected rows. Other processes can then read these rows, but cannot change the table entries. You should always lock the selected rows when you are going to make changes (**SQL SELECT ... FOR UPDATE**).
- 3 Read rows from the result set, modify and/or add new rows:
 - Adopt one row of the result set into the Q parameters of your NC program (**SQL FETCH...**)
 - Prepare changes in the Q parameters and transfer to a row in the result set (**SQL UPDATE...**)
 - Prepare new table row in the Q parameters and transfer as a new row to the result set (**SQL INSERT...**)
- 4 Conclude transaction:
 - If changes/insertions were made, the data from the result set is placed in the table (file). The data is now saved in the file. Any locks are canceled, and the result set is released (**SQL COMMIT...**).
 - If table entries were **not** changed or inserted (only read access), any locks are canceled and the result set is released (**SQL ROLLBACK... WITHOUT INDEX**).

Multiple transactions can be edited at the same time.



You must conclude a transaction, even if it consists solely of read accesses. Only this guarantees that changes/insertions are not lost, that locks are canceled, and that result sets are released.



Result set

The selected rows are numbered in ascending order within the result set, starting from 0. This numbering is referred to as the **index**. The index is used for read and write accesses, enabling a row of the result set to be specifically addressed.

It can often be advantageous to sort the rows in the result set. Do this by specifying the table column containing the sorting criteria. Also select ascending or descending order (**SQL SELECT ... ORDER BY ...**).

The selected rows that were transferred to the result set are addressed with the **HANDLE**. All following SQL commands use the handle to refer to this "set of selected columns and rows."

When concluding a transaction, the handle is released (**SQL COMMIT...** or **SQL ROLLBACK...**). It is then no longer valid.

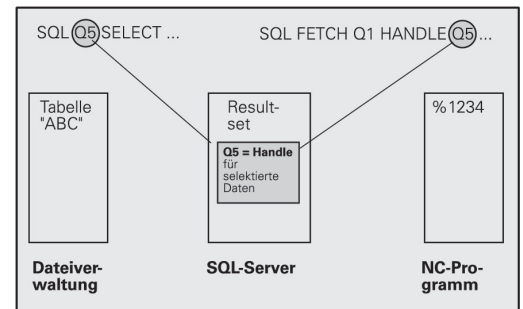
You can edit more than one result set at the same time. The SQL server assigns a new handle for each "Select" command.

"Binding" Q parameters to columns

The NC program does not have direct access to the table entries in the result set. The data must be transferred in Q parameters. In the other direction, the data is first prepared in the Q parameters and then transferred to the result set.

Specify with **SQL BIND ...** which table columns are mapped to which Q parameters. The Q parameters are "bound" (assigned) to the columns. Columns that are not bound to Q parameters are not included in the read-/write-processes.


If a new table row is generated with **SQL INSERT...**, the columns not bound to Q parameters are filled with default values.



9.9

Accessing tables with SQL commands

Programming SQL commands



This function can only be programmed if you have entered the code number 555343.

Program SQL commands in the **Programming** mode:

SPEC FCT

▶ Press the **SPEC FCT** key

PROGRAM FUNCTIONS

▶ Press the **PROGRAM FUNCTIONS** soft key.

▶

▶ Shift the soft-key row

SQL

▶ To select the MOD functions, press **SQL**
 ▶ Select an SQL command via soft key (see overview) or press the **SQL EXECUTE** soft key and program the SQL command

Overview of the soft keys

Soft key	Function
SQL BIND	SQL BIND Bind a Q parameter to a table column
SQL SELECT	SQL SELECT Select table rows
SQL EXECUTE	SQL EXECUTE Program a Select command
SQL FETCH	SQL FETCH Read table rows from the result set and save them in Q parameters
SQL ROLLBACK	SQL ROLLBACK <ul style="list-style-type: none"> ■ If INDEX is not programmed: Discard any changes/insertions and conclude the transaction. ■ If INDEX is programmed: The indexed row remains in the result set. All other rows are deleted from the result set. The transaction is not concluded.
SQL COMMIT	SQL COMMIT Transfer table rows from the result set into the table and conclude the transaction.
SQL UPDATE	SQL UPDATE Save data from the Q parameters in an existing table row in the result set
SQL INSERT	SQL INSERT Save data from the Q parameters in a new table row in the result set

SQL BIND

SQL BIND binds a Q parameter to a table column. The SQL commands "Fetch," "Update" and "Insert" evaluate this binding (assignment) during data transfer between the result set and the NC program.

An **SQL BIND** command without a table or column name cancels the binding. Binding remains effective at most until the end of the NC program or subprogram.



- You can program any number of bindings. Read and write processes only take into account the columns that were entered in the "Select" command.
- **SQL BIND...** must be programmed **before** "Fetch," "Update" or "Insert" commands are programmed. You can program a "Select" command without a preceding "Bind" command.
- If in the "Select" command you include columns for which no binding is programmed, an error occurs during read/write processes (program interrupt).

SQL
BIND

- ▶ **Parameter no. for result:** Q parameter that is bound (assigned) to the table column.
- ▶ **Database: Column name:** Enter the table name and column name separated by a . (period)
Table name: Synonym or path and file name of this table. The synonym is entered directly, whereas the path and file name are entered in single quotation marks.
Column designation: Designation of the table column as given in the configuration data

Bind a Q parameter to a table column

```
11SQL BIND Q881
   "TAB_EXAMPLE.MEAS_NO"
```

```
12SQL BIND Q882
   "TAB_EXAMPLE.MEAS_X"
```

```
13SQL BIND Q883
   "TAB_EXAMPLE.MEAS_Y"
```

```
14SQL BIND Q884
   "TAB_EXAMPLE.MEAS_Z"
```

Cancel binding

```
91 SQL BIND Q881
```

```
92 SQL BIND Q882
```

```
93 SQL BIND Q883
```

```
94 SQL BIND Q884
```


9.9

Accessing tables with SQL commands

SQL SELECT

SQL SELECT selects table rows and transfers them to the result set. The SQL server places the data in the result set row-by-row. The rows are numbered in ascending order, starting from 0. This row number, called the **INDEX**, is used in the SQL commands "Fetch" and "Update."

Enter the selection criteria in the **SQL SELECT...WHERE...** function. This lets you restrict the number of rows to be transferred. If you do not use this option, all rows in the table are loaded.

Enter the sorting criteria in the **SQL SELECT...ORDER BY...** function. Enter the column designation and the keyword for ascending/ descending order. If you do not use this option, the rows are placed in random order.

Lock out the selected rows for other applications with the **SQL SELECT...FOR UPDATE** function. Other applications can continue to read these rows, but cannot change them. We strongly recommend using this option if you are making changes to the table entries.

Empty result set: If no rows match the selection criteria, the SQL server returns a valid handle but no table entries.



- ▶ **Parameter no. for result:** Q parameter for the handle. The SQL server returns the handle for the group of columns and rows selected with the current "Select" command.
With an error (selection could not be executed) the SQL server returns a 1. Code 0 identifies an invalid handle.
- ▶ **Data bank: SQL command text:** with the following elements:
 - **SELECT** (keyword):
Name of the SQL command, names of the table columns to be transferred. Separate column names with a , (comma) (see examples). Q parameters must be bound to all columns entered here.
 - **FROM** table name:
Synonym or path and file name of this table. The synonym is entered directly: the path name and table name are entered in single quotation marks (see examples of the SQL command); names of the table columns to be transferred—separate several columns by a comma (see examples). Q parameters must be bound to all columns entered here.

Select all table rows

```
11SQL BIND Q881
   "TAB_EXAMPLE.MEAS_NO"
12SQL BIND Q882
   "TAB_EXAMPLE.MEAS_X"
13SQL BIND Q883
   "TAB_EXAMPLE.MEAS_Y"
14SQL BIND Q884
   "TAB_EXAMPLE.MEAS_Z"
...
20SQL Q5 "SELECT
   MEAS_NO,MEAS_X,MEAS_Y,
   MEAS_Z FROM TAB_EXAMPLE"
```

Selection of table rows with the WHERE function

```
...
20SQL Q5 "SELECT
   MEAS_NO,MEAS_X,MEAS_Y,
   MEAS_Z FROM TAB_EXAMPLE
   WHERE MEAS_NO<20"
```

Selection of table rows with the WHERE function and Q parameters

```
...
20SQL Q5 "SELECT
   MEAS_NO,MEAS_X,MEAS_Y,
   MEAS_Z FROM TAB_EXAMPLE
   WHERE MEAS_NO==:'Q11'"
```

Table name defined with path and file name

```
...
20SQL Q5 "SELECT
   MEAS_NO,MEAS_X,MEAS_Y,
   MEAS_Z FROM 'V:\TABLE
   \TAB_EXAMPLE' WHERE
   MEAS_NO<20"
```


- Optional:
WHERE selection criteria: A selection criterion consists of a column name, condition (see table) and comparator. Link several selection criteria with logical AND or OR. Program the comparative value directly or with a Q parameter. A Q parameter is introduced with a colon and placed in single quotation marks (see example).
- Optional:
ORDER BY column name **ASC** for ascending sorting, or **ORDER BY** column name **DESC** for descending sorting. If you program neither ASC nor DESC, ascending sorting is executed by default. The TNC places the selected rows in the indicated column.
- Optional:
FOR UPDATE (keyword): The selected rows are locked against write-accesses from other processes.

Condition	Programming
Equal to	= ==
Not equal to	!= <>
Less than	<
Less than or equal to	<=
Greater than	>
Greater than or equal to	>=
Linking multiple conditions:	
Logical AND	AND
Logical OR	OR

9.9 Accessing tables with SQL commands

SQL FETCH

SQL FETCH reads the row addressed with **INDEX** from the result set, and places the table entries in the bound (assigned) Q parameters. The result set is addressed with the **HANDLE**.
SQL FETCH takes into account all columns entered in the "Select" command.



- ▶ **Parameter no. for result:** Q parameter, in which the SQL server has reported the result:
0: No error occurred
1: Error occurred (incorrect handle or index too large)
- ▶ **Database: SQL access ID:** Q parameter with the **handle** for identifying the result set (also see **SQL SELECT**).
- ▶ **Database: Index to SQL result:** Line number within the result set. The table entries of this row are read and are transferred into the bound Q parameters. If you do not enter an index, the first row is read (n=0).
Either enter the row number directly or program the Q parameter containing the index

Row number is transferred in a Q parameter

```
11SQL BIND Q881
   "TAB_EXAMPLE.MEAS_NO"
12SQL BIND Q882
   "TAB_EXAMPLE.MEAS_X"
13SQL BIND Q883
   "TAB_EXAMPLE.MEAS_Y"
14SQL BIND Q884
   "TAB_EXAMPLE.MEAS_Z"
...
20SQL Q5 "SELECT
   MEAS_NO,MEAS_X,MEAS_Y,
   MEAS_Z FROM TAB_EXAMPLE"
...
30 SQL FETCH Q1HANDLE Q5 INDEX
   +Q2
```

Row number is programmed directly

```
...
30 SQL FETCH Q1HANDLE Q5 INDEX5
```


SQL UPDATE

SQL UPDATE transfers the data prepared in the Q parameters into the row of the result set addressed with **INDEX**. The existing row in the result set is completely overwritten.

SQL UPDATE takes into account all columns entered in the "Select" command.

SQL
UPDATE

- ▶ **Parameter no. for result:** Q parameter, in which the SQL server has reported the result:
0: No error occurred
1: Error occurred (incorrect handle, index too large, value outside of value range or incorrect data format)
- ▶ **Database: SQL access ID:** Q parameter with the **handle** for identifying the result set (also see **SQL SELECT**).
- ▶ **Database: Index to SQL result:** Line number within the result set. The table entries prepared in the Q parameters are written to this row. If you do not enter an index, the first row is written to (n=0). Either enter the row number directly or program the Q parameter containing the index

Row number is programmed directly

...

40 SQL UPDATEQ1 HANDLE Q5 INDEX5

SQL INSERT

SQL INSERT generates a new row in the result set and transfers the data prepared in the Q parameters into the new row.

SQL INSERT takes into account all columns entered in the "Select" command. Table columns not entered in the "Select" command are filled with default values.

SQL
INSERT

- ▶ **Parameter no. for result:** Q parameter, in which the SQL server has reported the result:
0: No error occurred
1: Error occurred (incorrect handle, value outside of value range or incorrect data format)
- ▶ **Database: SQL access ID:** Q parameter with the **handle** for identifying the result set (also see **SQL SELECT**).

Row number is transferred in a Q parameter

11SQL BIND Q881
"TAB_EXAMPLE.MEAS_NO"

12SQL BIND Q882
"TAB_EXAMPLE.MEAS_X"

13SQL BIND Q883
"TAB_EXAMPLE.MEAS_Y"

14SQL BIND Q884
"TAB_EXAMPLE.MEAS_Z"

...

20SQL Q5 "SELECT
MEAS_NO,MEAS_X,MEAS_Y,
MEAS_Z FROM TAB_EXAMPLE"

...

40 SQL INSERTQ1 HANDLE Q5

9.9 Accessing tables with SQL commands

SQL COMMIT

SQL COMMIT transfers all rows in the result set back to the table. A lock set with **SELECT...FOR UPDATE** is canceled.

The handle given in the **SQL SELECT** command loses its validity.



- ▶ **Parameter no. for result:** Q parameter, in which the SQL server has reported the result:
0: No error occurred
1: Error occurred (incorrect handle or equal entries in columns requiring unique entries)
- ▶ **Database: SQL access ID:** Q parameter with the **handle** for identifying the result set (also see **SQL SELECT**).

11	SQL BIND Q881
	"TAB_EXAMPLE.MEAS_NO"
12	SQL BIND Q882
	"TAB_EXAMPLE.MEAS_X"
13	SQL BIND Q883
	"TAB_EXAMPLE.MEAS_Y"
14	SQL BIND Q884
	"TAB_EXAMPLE.MEAS_Z"
...	
20	SQL Q5 "SELECT
	MEAS_NO,MEAS_X,MEAS_Y,
	MEAS_Z FROM TAB_EXAMPLE"
...	
30	SQL FETCH Q1HANDLE Q5 INDEX
	+Q2
...	
40	SQL UPDATEQ1 HANDLE Q5 INDEX
	+Q2
...	
50	SQL COMMITQ1 HANDLE Q5

SQL ROLLBACK

How **SQL ROLLBACK** is executed depends on whether **INDEX** is programmed:

- If **INDEX** is not programmed: The result set is **not** written back to the table (any changes/insertions are discarded). The transaction is closed and the handle given in the **SQL SELECT** command loses its validity. Typical application: Ending a transaction solely containing read-accesses.
- If **INDEX** is programmed: The indexed row remains. All other rows are deleted from the result set. The transaction is **not** concluded. A lock set with **SELECT...FOR UPDATE** remains for the indexed row. For all other rows it is reset.



- ▶ **Parameter no. for result:** Q parameter, in which the SQL server has reported the result:
0: No error occurred
1: Error occurred (incorrect handle)
- ▶ **Database: SQL access ID:** Q parameter with the **handle** for identifying the result set (also see **SQL SELECT**).
- ▶ **Database: Index to SQL result:** Line that is to remain in the result set. Either enter the row number directly or program the Q parameter containing the index














11	SQL BIND Q881
	"TAB_EXAMPLE.MEAS_NO"
12	SQL BIND Q882
	"TAB_EXAMPLE.MEAS_X"
13	SQL BIND Q883
	"TAB_EXAMPLE.MEAS_Y"
14	SQL BIND Q884
	"TAB_EXAMPLE.MEAS_Z"
...	
20	SQL Q5 "SELECT
	MEAS_NO,MEAS_X,MEAS_Y,
	MEAS_Z FROM TAB_EXAMPLE"
...	
30	SQL FETCH Q1HANDLE Q5 INDEX
	+Q2
...	
50	SQL ROLLBACKQ1 HANDLE Q5

9.10 Entering formulas directly

Entering formulas

You can enter mathematical formulas that include several operations directly into the part program by soft key.

Press the **FORMULA** soft key to call the mathematical functions. The TNC 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. $Q12 = Q1 * (Q2 + Q3)$
	Square e. g. $Q15 = SQ 5$
	Square root e. g. $Q22 = SQRT 25$
	Sine of an angle e. g. $Q44 = SIN 45$
	Cosine of an angle e. g. $Q45 = COS 45$
	Tangent of an angle e. g. $Q46 = TAN 45$
	Arc sine Inverse function of the sine; determine the angle from the ratio of the opposite side to the hypotenuse e.g. $Q10 = ASIN 0.75$
	Arc cosine Inverse function of the cosine; determine the angle from the ratio of the adjacent side to the hypotenuse e. g. $Q11 = ACOS Q40$

Programming: Q Parameters

9.10 Entering formulas directly

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 (3.4159) e.g. Q15 = PI
LN	Natural logarithm (LN) 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 (multiplication by -1) e.g. Q2 = NEG Q1
INT	Truncate digits after the decimal point Form an integer e.g. Q3 = INT Q42
ABS	Absolute value of a number e. g. Q4 = ABS Q22
FRAC	Truncate digits before the decimal point Form a fraction e.g. Q5 = FRAC Q23
SGN	Check algebraic sign of a number e.g. Q12 = SGN Q50 When return value Q12 = 1, then Q50 >= 0 When return value Q12 = -1, then Q50 < 0
%	Calculate modulo value (division remainder) e. g. Q12 = 400 % 360 Result: Q12 = 40

Rules for formulas

Mathematical formulas are programmed according to the following rules:

Higher-level operations are performed first

12 Q1 = 5 * 3 + 2 * 10 = 35

- 1 Calculation $5 * 3 = 15$
- 2 Calculation $2 * 10 = 20$
- 3 Calculation $15 + 20 = 35$

or

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

$$a * (b + c) = a * b + a * c$$

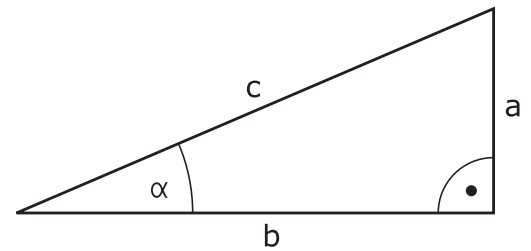
Programming: Q Parameters

9.10 Entering formulas directly

Programming example

Calculate an angle with the arc tangent from the opposite side (Q12) and adjacent side (Q13); then store in Q25.

- ▶ Select the formula entering function: Press the **Q** key and the **FORMULA** soft key, or use the shortcut:
- ▶ Press the **Q** key on the ASCII keyboard.



PARAMETER NUMBER FOR RESULT?

- ▶ Enter parameter number **25** and press the **ENT** key.
- ▶ Shift the soft-key row and select the arc tangent function
- ▶ Shift the soft-key row and open the parentheses
- ▶ Enter Q parameter number **12**
- ▶ Select division
- ▶ Enter Q parameter number **13**
- ▶ Close parentheses and conclude formula entry

Example NC block

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


9.11 String parameters

String processing functions

You can use the **QS** parameters to create variable character strings. You can output such character strings for example through the **FN 16:F-PRINT** function to create variable logs.

You can assign a linear sequence of characters (letters, numbers, special characters and spaces) up to a length of 255 characters to a string parameter. You can also check and process the assigned or imported values by using the functions described below. As in Q-parameter programming, you can use a total of 2000 QS parameters (see "Principle and overview of functions", page 298).

The **STRING FORMULA** and **FORMULA** Q-parameter functions contain various functions for processing the string parameters.

Soft key	STRING FORMULA functions	Page
STRING	Assigning string parameters	346
	Chain-linking string parameters	346
TOCHAR	Converting a numerical value to a string parameter	347
SUBSTR	Copy a substring from a string parameter	348
Soft key	FORMULA string functions	Page
TONUMB	Converting a string parameter to a numerical value	349
INSTR	Checking a string parameter	350
STRLEN	Finding the length of a string parameter	351
STRCOMP	Compare alphabetic priority	352



When you use a **STRING FORMULA**, 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.

9.11 String parameters

Assigning string parameters

You have to assign a string variable before you use it. Use the **DECLARE STRING** command to do so.

SPEC
FCT

- ▶ Show the soft-key row with special functions

PROGRAM
FUNCTIONS

- ▶ Open the function menu

STRING
FUNCTIONS

- ▶ Select string functions

DECLARE
STRING

- ▶ Select the **DECLARE STRING** function

Example NC block

```
37 DECLARE STRING QS10 = "WORKPIECE"
```

Chain-linking string parameters

With the concatenation operator (string parameter || string parameter) you can make a chain of two or more string parameters.

SPEC
FCT

- ▶ Show the soft-key row with special functions

PROGRAM
FUNCTIONS

- ▶ Open the function menu

STRING
FUNCTIONS

- ▶ Select string functions

STRING
FORMULA

- ▶ Select the **STRING FORMULA** function
- ▶ Enter the number of the string parameter in which the TNC 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 TNC displays the concatenation symbol ||
- ▶ Confirm your entry with the **ENT** key
- ▶ Enter the number of the string parameter in which the **second** substring is saved. Confirm with the **ENT** key
- ▶ Repeat the process until you have selected all the required substrings. Conclude with the **END** key

Example: QS10 is to include the complete text of QS12, QS13 and QS14

```
37 QS10 = QS12 || QS13 || QS14
```

Parameter contents:

- **QS12: Workpiece**
- **QS13: Status:**
- **QS14: Scrap**
- **QS10: Workpiece Status: Scrap**

Converting a numerical value to a string parameter

With the **TOCHAR** function, the TNC converts a numerical value to a string parameter. This enables you to chain numerical values with string variables.

- | | |
|---|--|
| <div style="border: 1px solid black; padding: 2px; width: fit-content; margin-bottom: 10px;">SPEC
FCT</div> <div style="border: 1px solid black; padding: 2px; width: fit-content; margin-bottom: 10px;">PROGRAM
FUNCTIONS</div> <div style="border: 1px solid black; padding: 2px; width: fit-content; margin-bottom: 10px;">STRING
FUNCTIONS</div> <div style="border: 1px solid black; padding: 2px; width: fit-content; margin-bottom: 10px;">STRING
FORMULA</div> <div style="border: 1px solid black; padding: 2px; width: fit-content;">TOCHAR</div> | <ul style="list-style-type: none"> ▶ Show the soft-key row with special functions
 ▶ Open the function menu
 ▶ Select string functions
 ▶ Select the STRING FORMULA function
 ▶ Select the function for converting a numerical value to a string parameter ▶ Enter the number or the desired Q parameter to be converted, and confirm with the ENT key ▶ If desired, enter the number of decimal places that the TNC should convert, and confirm with the ENT key ▶ Close the parenthetical expression with the ENT key and confirm your entry with the END key |
|---|--|

Example: Convert parameter Q50 to string parameter QS11, use 3 decimal places

```
37 QS11 = TOCHAR ( DAT+Q50 DECIMALS3 )
```


9.11 String parameters

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

- ▶ Select string functions

STRING
FORMULA

- ▶ Select the **STRING FORMULA** function
- ▶ Enter the number of the string parameter in which the TNC is to save the copied 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



Remember that the first character of a text sequence starts internally with the zeroth place.

Example: A four-character substring (LEN4) is read from the string parameter QS10 beginning with the third character (BEG2)

```
37 QS13 = SUBSTR ( SRC_QS10 BEG2 LEN4 )
```


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 must contain only one numerical value. Otherwise the TNC will output an error message.

Q

- ▶ Select Q-parameter functions

FORMULA

- ▶ Select the **FORMULA** function
- ▶ Enter the number of the parameter in which the TNC 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 Q parameter to be converted, and confirm with the **ENT** key
- ▶ Close the parenthetical expression with the **ENT** key and confirm your entry with the **END** key

Example: Convert string parameter QS11 to a numerical parameter Q82










```
37 Q82 = TONUMB ( SRC_QS11 )
```


9 Programming: Q Parameters

9.11 String parameters

Checking a string parameter

The **INSTR** function checks whether a string parameter is contained in another string parameter.

-  ▶ Select Q-parameter functions
-  ▶ Select the **FORMULA** function
-  ▶ Enter the number of the Q parameter for the result and confirm with the **ENT** key. The TNC saves in the parameter the position at which the sought-after text begins
-  ▶ Shift the soft-key row
-  ▶ 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, and confirm with the ENT key
-  ▶ Enter the number of the place starting from which the TNC is to 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



Remember that the first character of a text sequence starts internally with the zeroth place.

If the TNC 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 is found in more than one place, the TNC returns the first place at which it finds the substring.

Example: Search through QS10 for the text saved in parameter QS13. Begin the search at the third place.

```
37 Q50 = INSTR ( SRC_QS10 SEA_QS13 BEG2 )
```


Finding the length of a string parameter

The **STRLEN** function returns the length of the text saved in a selectable string parameter.

-  ▶ Select Q-parameter functions
-  ▶ Select the **FORMULA** function
-  ▶ Enter the number of the Q parameter in which the TNC is to save the ascertained string length. Confirm with the **ENT** key
-  ▶ Shift the soft-key row
-  ▶ Select the function for finding the text length of a string parameter
-  ▶ Enter the number of the QS parameter whose length the TNC is to ascertain, and confirm with the ENT key
-  ▶ Close the parenthetical expression with the **ENT** key and confirm your entry with the **END** key

Example: Find the length of QS15






```
37 Q52 = STRLEN ( SRC_QS15 )
```


9 Programming: Q Parameters

9.11 String parameters

Comparing alphabetic sequence

The **STRCOMP** function compares string parameters for alphabetic priority.

-  ▶ Select Q-parameter functions
-  ▶ Select the **FORMULA** function
-  ▶ Enter the number of the Q parameter in which the TNC is to save the result of comparison. Confirm with the **ENT** key
-  ▶ Shift the soft-key row
- 
 - ▶ Select the function for comparing string parameters
 - ▶ Enter the number of the first QS parameter to be compared, and confirm with the **ENT** key
 - ▶ Enter the number of the second QS parameter to be compared, and confirm with the **ENT** key
 - ▶ Close the parenthetical expression with the **ENT** key and confirm your entry with the **END** key



The TNC returns the following results:

- **0**: The compared QS parameters are identical
- **-1**: The first QS parameter **precedes** the second QS parameter alphabetically
- **+1**: The first QS parameter **follows** the second QS parameter alphabetically





Example: QS12 and QS14 are compared for alphabetic priority

```
37 Q52 = STRCOMP ( SRC_QS12 SEA_QS14 )
```


Reading out machine parameters

Use the **CFGREAD** function to read out TNC machine parameters as numerical values or as strings.

In order to read out a machine parameter, you must use the TNC'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 assigned)	CH_NC
	Entity	Parameter object (the name starts with "Cfg...")	CfgGeoCycle
	Attribute	Name of the machine parameter	displaySpindleErr
	Index	List index of a machine parameter (if assigned)	[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. To display the actual system names of the parameters, press the key for the screen layout key and then the **SHOW SYSTEM NAME** soft key. Follow the same procedure to return to the standard display.

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

9.11

String parameters

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

STRING
FORMULA

press the Q key.

Select the **STRING FORMULA** function

Enter the number of the string parameter in which the TNC is to save the machine parameter. Confirm with the **ENT** key

Select the CFGREAD function

Enter the numbers of the string parameters for the key, entity and attribute, then confirm with the **ENT** key

Enter the number for the index, or skip the dialog with NO ENT, whichever applies

Close the parenthetical expression with the **ENT** key and confirm your entry with the **END** key
- Example: Read as a string the axis designation of the fourth axis
- Parameter settings in the configuration editor
- DisplaySettings

CfgDisplayData

axisDisplayOrder

[0] to [5]
- | | |
|--|--|
| 14 DECLARE STRINGQS11 = "" | Assign string parameter for key |
| 15 DECLARE STRINGQS12 = "CFGDISPLAYDATA" | Assign string parameter for entity |
| 16 DECLARE STRINGQS13 = "AXISDISPLAYORDER" | Assign string parameter for parameter name |
| 17 QS1 =
CFGREAD(KEY_QS11 TAG_QS12 ATR_QS13 IDX3) | Read out machine parameter |
- 354

TNC 640 | User's Manual

HEIDENHAIN Conversational Programming | 1/2015

Reading a numerical value of a machine parameter

Store the value of a machine parameter as a numerical value in a Q parameter:

- Q

FORMULA

- ▶ Select Q-parameter functions
 - ▶ Select the FORMULA function
 - ▶ Enter the number of the Q parameter in which the TNC is to save the machine parameter. Confirm with the **ENT** key
 - ▶ Select the CFGREAD function
 - ▶ Enter the numbers of the string parameters for the key, entity and attribute, then confirm with the **ENT** key
 - ▶ Enter the number for the index, or skip the dialog with NO ENT, whichever applies
 - ▶ Close the parenthetical expression with the **ENT** key and confirm your entry with the **END** key

Example: Read overlap factor as Q parameter

Parameter settings in the configuration editor

```
ChannelSettings
CH_NC
    CfgGeoCycle
        pocketOverlap
```

14 DECLARE STRINGQ\$11 = "CH_NC"	Assign string parameter for key
15 DECLARE STRINGQ\$12 = "CFGGEOCYCLE"	Assign string parameter for entity
16 DECLARE STRINGQ\$13 = "POCKETOVERLAP"	Assign string parameter for parameter name
17 Q50 = CFGREAD(KEY_Q\$11 TAG_Q\$12 ATR_Q\$13)	Read out machine parameter


9.12 Preassigned Q parameters

9.12 Preassigned Q parameters

The Q parameters Q100 to Q199 are assigned values by the TNC. The following types of information are assigned to Q parameters:

- Values from the PLC
- Tool and spindle data
- Data on operating status
- Results of measurements from touch probe cycles etc.

The TNC saves the values for the preassigned Q parameters Q108, Q114 and Q115 to Q117 in the unit of measure used by the active program.



Do not use preassigned Q parameters (or QS parameters) between **Q100** and **Q199 (QS100 and QS199)** as calculation parameters in NC programs. Otherwise you might receive undesired results.


Values from the PLC: Q100 to Q107

The TNC uses the parameters Q100 to Q107 to transfer values from the PLC to an NC program.

Active tool radius: Q108

The active value of the tool radius is assigned to Q108. Q108 is calculated from:

- Tool radius R (tool table or **TOOL DEF** block)
- Delta value DR from the tool table
- Delta value DR from the **TOOL CALL** block



The TNC 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 overlap factor for pocket milling is assigned to Q112.

Unit of measurement for dimensions in the program: Q113

During nesting with PGM CALL, the value of the parameter Q113 depends on the dimensional data of the program from which the other programs are called.

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

Tool length: Q114

The current value for the tool length is assigned to Q114.



The TNC remembers the current tool length even if the power is interrupted.

Programming: Q Parameters

9.12 Preassigned Q parameters

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
V. axis Machine-dependent	Q119

Deviation between actual value and nominal value during automatic tool measurement with the TT 130

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

Tilting the working plane with mathematical angles: rotary axis coordinates calculated by the TNC

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

Measurement results from touch probe cycles (see also User's Manual for Cycle Programming)

Measured actual values	Parameter value
Angle of a straight line	Q150
Center in reference axis	Q151
Center in minor axis	Q152
Diameter	Q153
Pocket length	Q154
Pocket width	Q155
Length of the axis selected in the cycle	Q156
Position of the centerline	Q157
Angle in the A axis	Q158
Angle in the B axis	Q159
Coordinate of the axis selected in the cycle	Q160
Measured deviation	Parameter value
Center in reference axis	Q161
Center in minor axis	Q162
Diameter	Q163
Pocket length	Q164
Pocket width	Q165
Measured length	Q166
Position of the centerline	Q167
Determined space angle	Parameter value
Rotation about the A axis	Q170
Rotation about the B axis	Q171
Rotation about the C axis	Q172
Workpiece status	Parameter value
Good	Q180
Rework	Q181
Scrap	Q182

9 Programming: Q Parameters

9.12 Preassigned Q parameters

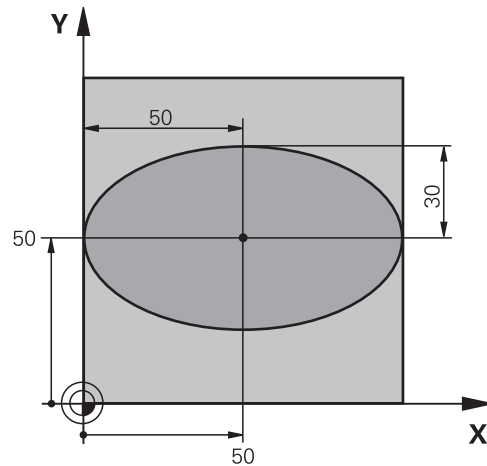
Tool measurement with the BLUM laser	Parameter value
Reserved	Q190
Reserved	Q191
Reserved	Q192
Reserved	Q193
Reserved for internal use	Parameter value
Marker for cycles	Q195
Marker for cycles	Q196
Marker for cycles (machining patterns)	Q197
Number of the last active measuring cycle	Q198
Status of tool measurement with TT	Parameter value
Tool within tolerance	Q199 = 0.0
Tool is worn (LTOL/RTOL is exceeded)	Q199 = 1.0
Tool is broken (LBREAK/RBREAK is exceeded)	Q199 = 2.0

9.13 Programming examples

Example: Ellipse

Program sequence

- The contour of the ellipse is approximated by many short lines (defined in Q7). The more calculation steps you define for the lines, the smoother the curve becomes.
- The milling direction is determined with the starting angle and end angle in the plane :
Machining direction is clockwise:
Starting angle > end angle
Machining direction is counterclockwise:
Starting angle < end angle
- The tool radius is not taken into account



0 BEGIN PGM ELLIPSE MM	
1 FN 0: Q1 = +50	Center in X axis
2 FN 0: Q2 = +50	Center in Y axis
3 FN 0: Q3 = +50	Semixaxis in X
4 FN 0: Q4 = +30	Semixaxis in Y
5 FN 0: Q5 = +0	Starting angle in the plane
6 FN 0: Q6 = +360	End angle in the plane
7 FN 0: Q7 = +40	Number of calculation steps
8 FN 0: Q8 = +0	Rotational position of the ellipse
9 FN 0: Q9 = +5	Milling depth
10 FN 0: Q10 = +100	Feed rate for plunging
11 FN 0: Q11 = +350	Feed rate for milling
12 FN 0: Q12 = +2	Set-up clearance for pre-positioning
13 BLK FORM 0.1 Z X+0 Y+0 Z-20	Definition of workpiece blank
14 BLK FORM 0.2 X+100 Y+100 Z+0	
15 TOOL CALL 1 Z S4000	Tool call
16 L Z+250 R0 FMAX	Retract the tool
17 CALL LBL 10	Call machining operation
18 L Z+100 R0 FMAX M2	Retract the tool, end program
19 LBL 10	Subprogram 10: Machining operation
20 CYCL DEF 7.0 DATUM SHIFT	Shift datum to center of ellipse
21 CYCL DEF 7.1 X+Q1	
22 CYCL DEF 7.2 Y+Q2	
23 CYCL DEF 10.0 ROTATION	Account for rotational position in the plane
24 CYCL DEF 10.1 ROT+Q8	
25 Q35 = (Q6 -Q5) / Q7	Calculate angle increment
26 Q36 = Q5	Copy starting angle

Programming: Q Parameters

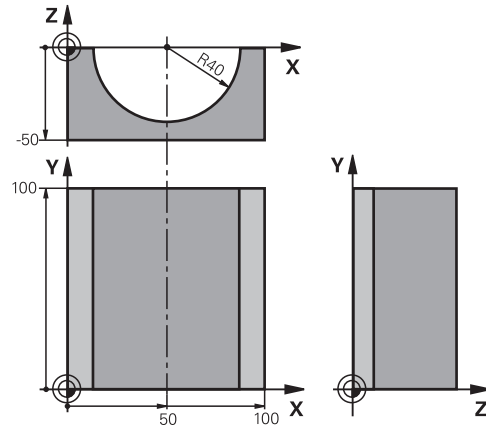
9.13 Programming examples

27 Q37 = 0	Set counter
28 Q21 = Q3 *COS Q36	Calculate X coordinate for starting point
29 Q22 = Q4 *SIN Q36	Calculate Y coordinate for starting point
30 L X+Q21 Y+Q22 R0 FMAX M3	Move to starting point in the plane
31 L Z+Q12 R0 FMAX	Pre-position in spindle axis to set-up clearance
32 L Z-Q9 R0 FQ10	Move to working depth
33 LBL 1	
34 Q36 = Q36 +Q35	Update the angle
35 Q37 = Q37 +1	Update the counter
36 Q21 = Q3 *COS Q36	Calculate the current X coordinate
37 Q22 = Q4 *SIN Q36	Calculate the current Y coordinate
38 L X+Q21 Y+Q22 R0 FQ11	Move to next point
39 FN 12: IF +Q37 LT +Q7 GOTO LBL 1	Unfinished? If not finished, return to LBL 1
40 CYCL DEF 10.0 ROTATION	Reset the rotation
41 CYCL DEF 10.1 ROT+0	
42 CYCL DEF 7.0 DATUM SHIFT	Reset the datum shift
43 CYCL DEF 7.1 X+0	
44 CYCL DEF 7.2 Y+0	
45 L Z+Q12 R0 FMAX	Move to set-up clearance
46 LBL 0	End of subprogram
47 END PGM ELLIPSE MM	

Example: Concave cylinder machined with spherical cutter

Program sequence

- This program functions only with a spherical 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



0 BEGIN PGM CYLIN MM	
1 FN 0: Q1 = +50	Center in X axis
2 FN 0: Q2 = +0	Center in Y axis
3 FN 0: Q3 = +0	Center in Z axis
4 FN 0: Q4 = +90	Starting angle in space (Z/X plane)
5 FN 0: Q5 = +270	End angle in space (Z/X plane)
6 FN 0: Q6 = +40	Cylinder radius
7 FN 0: Q7 = +100	Length of the cylinder
8 FN 0: Q8 = +0	Rotational position in the X/Y plane
9 FN 0: Q10 = +5	Allowance for cylinder radius
10 FN 0: Q11 = +250	Feed rate for plunging
11 FN 0: Q12 = +400	Feed rate for milling
12 FN 0: Q13 = +90	Number of cuts
13 BLK FORM 0.1 Z X+0 Y+0 Z-50	Definition of workpiece blank
14 BLK FORM 0.2 X+100 Y+100 Z+0	
15 TOOL CALL 1 Z S4000	Tool call
16 L Z+250 R0 FMAX	Retract the tool
17 CALL LBL 10	Call machining operation
18 FN 0: Q10 = +0	Reset allowance
19 CALL LBL 10	Call machining operation
20 L Z+100 R0 FMAX M2	Retract the tool, end program

Programming: Q Parameters

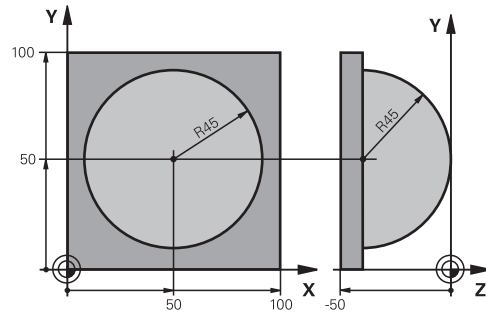
9.13 Programming examples

21 LBL 10	Subprogram 10: Machining operation
22 Q16 = Q6 -Q10 - Q108	Account for allowance and tool, based on the cylinder radius
23 FN 0: Q20 = +1	Set counter
24 FN 0: Q24 = +Q4	Copy starting angle in space (Z/X plane)
25 Q25 = (Q5 -Q4) / Q13	Calculate angle increment
26 CYCL DEF 7.0 DATUM SHIFT	Shift datum to center of cylinder (X axis)
27 CYCL DEF 7.1 X+Q1	
28 CYCL DEF 7.2 Y+Q2	
29 CYCL DEF 7.3 Z+Q3	
30 CYCL DEF 10.0 ROTATION	Account for rotational position in the plane
31 CYCL DEF 10.1 ROT+Q8	
32 L X+0 Y+0 R0 FMAX	Pre-position in the plane to the cylinder center
33 L Z+5 R0 F1000 M3	Pre-position in the spindle axis
34 LBL 1	
35 CC Z+0 X+0	Set pole in the Z/X plane
36 LP PR+Q16 PA+Q24 FQ11	Move to starting position on cylinder, plunge-cutting obliquely into the material
37 L Y+Q7 R0 FQ12	Longitudinal cut in Y+ direction
38 FN 1: Q20 = +Q20 + +1	Update the counter
39 FN 1: Q24 = +Q24 + +Q25	Update solid angle
40 FN 11: IF +Q20 GT +Q13 GOTO LBL 99	Finished? If finished, jump to end
41 LP PR+Q16 PA+Q24 FQ11	Move in an approximated "arc" for the next longitudinal cut
42 L Y+0 R0 FQ12	Longitudinal cut in Y– direction
43 FN 1: Q20 = +Q20 + +1	Update the counter
44 FN 1: Q24 = +Q24 + +Q25	Update solid angle
45 FN 12: IF +Q20 LT +Q13 GOTO LBL 1	Unfinished? If not finished, return to LBL 1
46 LBL 99	
47 CYCL DEF 10.0 ROTATION	Reset the rotation
48 CYCL DEF 10.1 ROT+0	
49 CYCL DEF 7.0 DATUM SHIFT	Reset the datum shift
50 CYCL DEF 7.1 X+0	
51 CYCL DEF 7.2 Y+0	
52 CYCL DEF 7.3 Z+0	
53 LBL 0	End of subprogram
54 END PGM CYLIN	

Example: Convex sphere machined with end mill

Program sequence

- This program requires an end mill.
- The contour of the sphere is approximated by many short lines (in the Z/X plane, defined in Q14). The smaller you define the angle increment, the smoother the curve becomes.
- You can determine the number of contour cuts through the angle increment in the plane (defined in Q18).
- The tool moves upward in three-dimensional cuts.
- The tool radius is compensated automatically



0 BEGIN PGM SPHERE MM	
1 FN 0: Q1 = +50	Center in X axis
2 FN 0: Q2 = +50	Center in Y axis
3 FN 0: Q4 = +90	Starting angle in space (Z/X plane)
4 FN 0: Q5 = +0	End angle in space (Z/X plane)
5 FN 0: Q14 = +5	Angle increment in space
6 FN 0: Q6 = +45	Sphere radius
7 FN 0: Q8 = +0	Starting angle of rotational position in the X/Y plane
8 FN 0: Q9 = +360	End angle of rotational position in the X/Y plane
9 FN 0: Q18 = +10	Angle increment in the X/Y plane for roughing
10 FN 0: Q10 = +5	Allowance in sphere radius for roughing
11 FN 0: Q11 = +2	Set-up clearance for pre-positioning in the spindle axis
12 FN 0: Q12 = +350	Feed rate for milling
13 BLK FORM 0.1 Z X+0 Y+0 Z-50	Definition of workpiece blank
14 BLK FORM 0.2 X+100 Y+100 Z+0	
15 TOOL CALL 1 Z S4000	Tool call
16 L Z+250 R0 FMAX	Retract the tool
17 CALL LBL 10	Call machining operation
18 FN 0: Q10 = +0	Reset allowance
19 FN 0: Q18 = +5	Angle increment in the X/Y plane for finishing
20 CALL LBL 10	Call machining operation
21 L Z+100 R0 FMAX M2	Retract the tool, end program
22 LBL 10	Subprogram 10: Machining operation
23 FN 1: Q23 = +Q11 + +Q6	Calculate Z coordinate for pre-positioning
24 FN 0: Q24 = +Q4	Copy starting angle in space (Z/X plane)
25 FN 1: Q26 = +Q6 + +Q108	Compensate sphere radius for pre-positioning
26 FN 0: Q28 = +Q8	Copy rotational position in the plane
27 FN 1: Q16 = +Q6 + -Q10	Account for allowance in the sphere radius
28 CYCL DEF 7.0 DATUM SHIFT	Shift datum to center of sphere
29 CYCL DEF 7.1 X+Q1	
30 CYCL DEF 7.2 Y+Q2	
31 CYCL DEF 7.3 Z-Q16	

Programming: Q Parameters

9.13 Programming examples

32 CYCL DEF 10.0 ROTATION	Account for starting angle of rotational position in the plane
33 CYCL DEF 10.1 ROT+Q8	
34 LBL 1	Pre-position in the spindle axis
35 CC X+0 Y+0	Set pole in the X/Y plane for pre-positioning
36 LP PR+Q26 PA+Q8 R0 FQ12	Pre-position in the plane
37 CC Z+0 X+Q108	Set pole in the Z/X plane, offset by the tool radius
38 L Y+0 Z+0 FQ12	Move to working depth
39 LBL 2	
40 LP PR+Q6 PA+Q24 FQ12	Move upward in an approximated "arc"
41 FN 2: Q24 = +Q24 - +Q14	Update solid angle
42 FN 11: IF +Q24 GT +Q5 GOTO LBL 2	Inquire whether an arc is finished. If not finished, return to LBL 2
43 LP PR+Q6 PA+Q5	Move to the end angle in space
44 L Z+Q23 R0 F1000	Retract in the spindle axis
45 L X+Q26 R0 FMAX	Pre-position for next arc
46 FN 1: Q28 = +Q28 + +Q18	Update rotational position in the plane
47 FN 0: Q24 = +Q4	Reset solid angle
48 CYCL DEF 10.0 ROTATION	Activate new rotational position
49 CYCL DEF 10.0 ROT+Q28	
50 FN 12: IF +Q28 LT +Q9 GOTO LBL 1	
51 FN 9: IF +Q28 EQU +Q9 GOTO LBL 1	Unfinished? If not finished, return to LBL 1
52 CYCL DEF 10.0 ROTATION	Reset the rotation
53 CYCL DEF 10.1 ROT+0	
54 CYCL DEF 7.0 DATUM SHIFT	Reset the datum shift
55 CYCL DEF 7.1 X+0	
56 CYCL DEF 7.2 Y+0	
57 CYCL DEF 7.3 Z+0	
58 LBL 0	End of subprogram
59 END PGM SPHERE MM	

10

**Programming:
Miscellaneous
functions**

Programming: Miscellaneous functions

10.1 Entering miscellaneous functions M and STOP

10.1 Entering miscellaneous functions M and STOP

Fundamentals

With the TNC'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



The machine tool builder may add some M functions that are not described in this User's Manual. Refer to your machine manual.

You can enter up to four M functions at the end of a positioning block or in a separate block. The TNC displays the following dialog question: **Miscellaneous function M ?**

You usually enter only the number of the M function in the programming dialog. Some M functions can be programmed with additional parameters. In this case, the dialog is continued for the parameter input.

In the **Manual Operation** and **El. Handwheel** modes of operation, the M functions are entered with the **M** soft key.



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.

M functions come into effect in the block in which they are called.

Some M functions are effective only in the block in which they are programmed. Unless the M function is only effective blockwise, either you must cancel it in a subsequent block with a separate M function, or it is automatically canceled by the TNC at the end of the program.

Entering an M function in a STOP block

If you program a **STOP** block, the program run or test run is interrupted at the block, for example for tool inspection. You can also enter an M function in a **STOP** block:

STOP

- ▶ To program an interruption of program run, press the **STOP** key.
- ▶ Enter a miscellaneous function **M**

Example NC blocks

87 STOP M6

10.2 M functions for program run inspection, spindle and coolant

Overview



The machine tool builder can influence the behavior of the miscellaneous functions described below. Refer to your machine manual.

M	Effect	Effective at block	Start	End
M0	Program STOP Spindle STOP			■
M1	Optional program STOP Spindle STOP if necessary Coolant OFF if necessary (not effective during Test Run, function determined by the machine tool builder)			■
M2	STOP program run Spindle STOP Coolant OFF Return jump to block 1 CLEAR status display (depending on machine parameter clearMode)			■
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			■

Programming: Miscellaneous functions

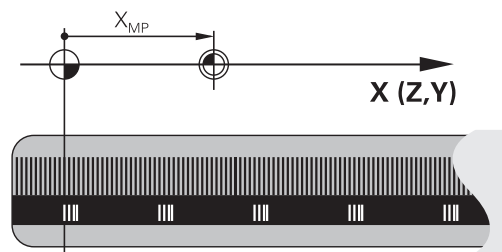
10.3 Miscellaneous functions for coordinate data

10.3 Miscellaneous functions for coordinate data

Programming machine-referenced coordinates: M91/M92

Scale reference point

On the scale, a reference mark indicates the position of the scale reference point.



Machine datum

The machine datum is required for the following tasks:

- Define the axis traverse limits (software limit switches)
- Approach machine-referenced positions (such as tool change positions)
- Set a workpiece datum

The distance in each axis from the scale reference point to the machine datum is defined by the machine tool builder in a machine parameter.

Standard behavior

The TNC references coordinates to the workpiece datum (see "Datum setting without a 3-D touch probe", page 543).

Behavior with M91 – Machine datum

If you want the coordinates in a positioning block to be referenced to the machine datum, end the block with M91.



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 TNC screen are referenced to the machine datum. Switch the display of coordinates in the status display to REF, see "Status displays", page 78.

Behavior with M92—Additional machine datum

In addition to the machine datum, the machine tool builder can also define an additional machine-based position as a reference point.

For each axis, the machine tool builder defines the distance between the machine datum and this additional machine datum. Refer to your machine manual.

If you want the coordinates in a positioning block to be based on the additional machine datum, end the block with M92.



Radius compensation remains the same in blocks that are programmed with M91 or M92. The tool length, however, is **not** compensated.

Effect

M91 and M92 are effective only in the blocks in which they are programmed.

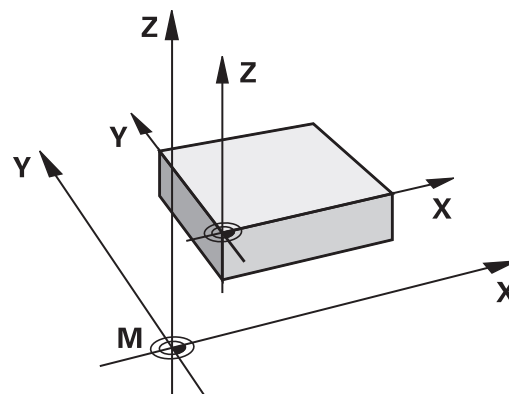
M91 and M92 take effect at the start of block.

Workpiece datum

If you want the coordinates to always be referenced to the machine datum, you can inhibit datum setting for one or more axes.

If datum setting is inhibited for all axes, the TNC no longer displays the SET DATUM soft key in the **Manual Operation** mode.

The figure shows coordinate systems with the machine datum 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 set datum, see "Showing the workpiece blank in the working space ", page 594.

Programming: Miscellaneous functions

10.3 Miscellaneous functions for coordinate data

Moving to positions in a non-tilted coordinate system with a tilted working plane: M130

Standard behavior with a tilted working plane

The TNC places the coordinates in the positioning blocks in the tilted coordinate system.

Behavior with M130

The TNC places coordinates in straight line blocks in the untilted coordinate system.

The TNC then positions the (tilted) tool to the programmed coordinates of the untilted system.



Danger of collision!

Subsequent positioning blocks or fixed cycles are carried out in a tilted coordinate system. This can lead to problems in fixed cycles with absolute pre-positioning.

The function M130 is allowed only if the tilted working plane function is active.

Effect

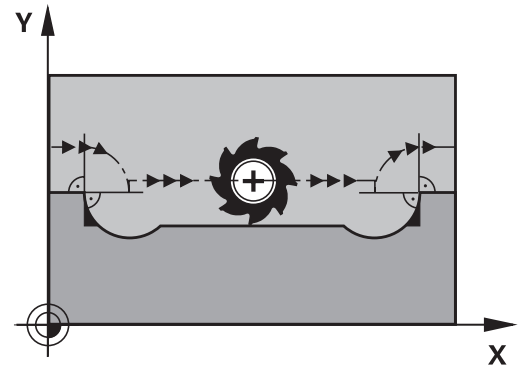
M130 functions blockwise in straight-line blocks without tool radius compensation.

10.4 Miscellaneous functions for path behavior

Machining small contour steps: M97

Standard behavior

The TNC inserts a transition arc at outside corners. If the contour steps are very small, however, the tool would damage the contour. In such cases the TNC interrupts program run and generates the error message "Tool radius too large."



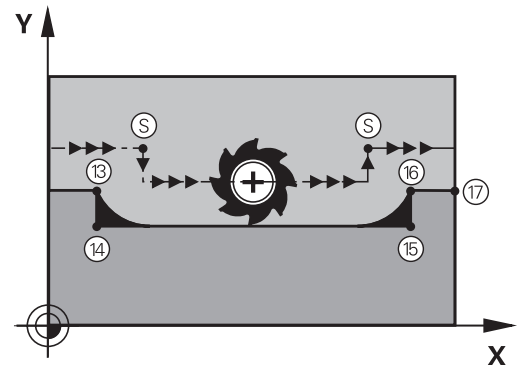
Behavior with M97

The TNC calculates the intersection of the contour elements—as at inside corners—and moves the tool over this point.

Program M97 in the same block as the outside corner.



Instead of **M97** you should use the much more powerful function **M120 LA**, see "Calculating the radius-compensated path in advance (LOOK AHEAD): M120 ", page 378.



Effect

M97 is effective only in the blocks in which it is programmed.



A corner machined with M97 will not be completely finished. You may wish to rework the contour with a smaller tool.

Example NC blocks

5 TOOL DEF L ... R+20	Large tool radius
...	
13 L X... Y... R... F... M97	Move to contour point 13
14 L IY-0.5 ... R... F...	Machine small contour step 13 to 14
15 L IX+100 ...	Move to contour point 15
16 L IY+0.5 ... R... F... M97	Machine small contour step 15 to 16
17 L X... Y...	Move to contour point 17

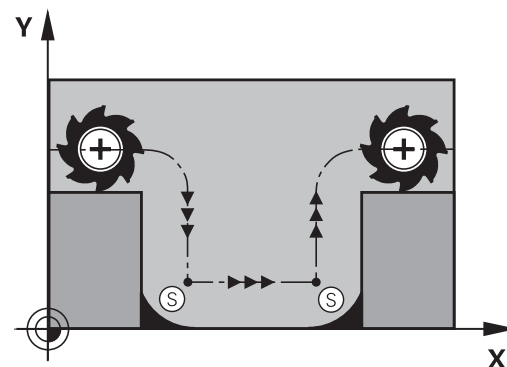
Programming: Miscellaneous functions

10.4 Miscellaneous functions for path behavior

Machining open contour corners: M98

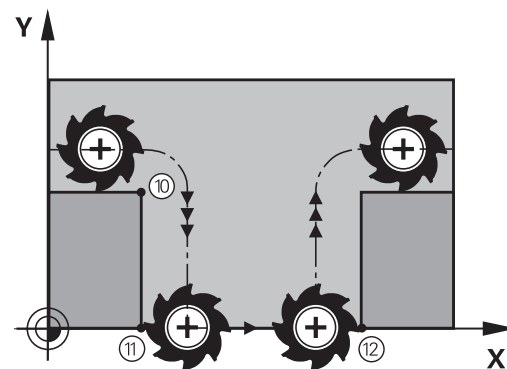
Standard behavior

The TNC 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 miscellaneous function M98, the TNC temporarily suspends radius compensation to ensure that both corners are completely machined:



Effect

M98 is effective only in the blocks in which it is programmed. M98 takes effect at the end of block.

Example NC blocks

Move to the contour points 10, 11 and 12 in succession:

```
10 L X... Y... RL F
```

```
11 L X... Y... M98
```

```
12 L IX+ ...
```


Feed rate factor for plunging movements: M103

Standard behavior

The TNC moves the tool at the last programmed feed rate, regardless of the direction of traverse.

Behavior with M103

The TNC 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%:

$$FZMAX = FPROG \times F\%$$

Programming M103

If you enter M103 in a positioning block, the TNC continues the dialog by asking you the factor F.

Effect

M103 becomes effective at the start of block.

To cancel M103, program M103 once again without a factor.



M103 is also effective in an active tilted working plane. The feed rate reduction is then effective during traverse in the negative direction of the **tilted** tool axis.

Example NC blocks

The feed rate for plunging is to be 20% of the feed rate in the plane.

...	Actual contouring feed rate (mm/min):
17 L X+20 Y+20 RL F500 M103 F20	500
18 L Y+50	500
19 L IZ-2.5	100
20 L IY+5 IZ-5	141
21 L IX+50	500
22 L Z+5	500

Programming: Miscellaneous functions

10.4 Miscellaneous functions for path behavior

Feed rate in millimeters per spindle revolution: M136

Standard behavior

The TNC moves the tool at the programmed feed rate F in mm/min

Behavior with M136



In inch-programs, M136 is not permitted in combination with the new alternate feed rate FU.
The spindle is not permitted to be controlled when M136 is active.

With M136, the TNC does not move the tool in mm/min, but rather at the programmed feed rate F in millimeters per spindle revolution. If you change the spindle speed by using the spindle override, the TNC changes the feed rate accordingly.

Effect

M136 becomes effective at the start of block.

You can cancel M136 by programming M137.

Feed rate for circular arcs: M109/M110/M111

Standard behavior

The TNC applies the programmed feed rate to the path of the tool center.

Behavior at circular arcs with M109

The TNC adjusts the feed rate for circular arcs at inside and outside contours so that the feed rate at the tool cutting edge remains constant.



Caution: Danger to the workpiece and tool!

On very small outside corners the TNC may increase the feed rate so much that the tool or workpiece can be damaged. Avoid **M109** with small outside corners.

Behavior at circular arcs with M110

The TNC keeps the feed rate constant for circular arcs at inside contours only. At outside contours, the feed rate is not adjusted.



If you define M109 or M110 before calling a machining cycle with a number greater than 200, the adjusted feed rate is also effective for circular arcs within these machining cycles. The initial state is restored after finishing or aborting a machining cycle.

Effect

M109 and M110 become effective at the start of block. To cancel M109 or M110, enter M111.

Programming: Miscellaneous functions

10.4 Miscellaneous functions for path behavior

Calculating the radius-compensated path in advance (LOOK AHEAD): M120

Standard behavior

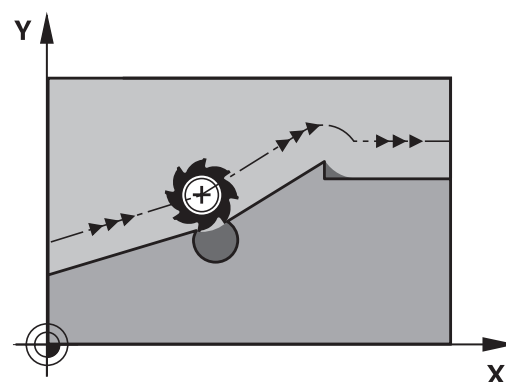
If the tool radius is larger than the contour step that is to be machined with radius compensation, the TNC interrupts program run and generates an error message. M97 (see "Machining small contour steps: M97", page 373) inhibits the error message, but this results in dwell marks and will also move the corner.

If the programmed contour contains undercut features, the tool may damage the contour.

Behavior with M120

The TNC checks radius-compensated paths for contour undercuts and tool path intersections, and calculates the tool path in advance from the current block. Areas of the contour that might be damaged by the tool are not machined (dark areas in figure). You can also use M120 to calculate the 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.

Use LA (Look Ahead) behind M120 to define the number of blocks (maximum: 99) that you want the TNC to calculate in advance. Note that the larger the number of blocks you choose, the higher the block processing time will be.



Input

If you enter M120 in a positioning block, the TNC continues the dialog for this block by asking you the number of blocks LA that are to be calculated in advance.

Effect

M120 must be located in an NC block that also contains radius compensation **RL** or **RR**. M120 is then effective from this block until

- radius compensation is canceled with **R0**
- M120 LA0 is programmed, or
- M120 is programmed without LA, or
- another program is called with **PGM CALL**
- the working plane is tilted with Cycle **19** or the PLANE function

M120 becomes effective at the start of 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 TNC will output an error message.
- When using the path functions **RND** and **CHF**, the blocks before and after **RND** or **CHF** must contain only coordinates in the working plane.
- If you want to approach the contour on a tangential path, you must use the function APPR LCT. The block with APPR LCT must contain only coordinates of the working plane
- If you want to depart the contour on a tangential path, use the function DEP LCT. The block with DEP LCT must contain only coordinates of the working plane
- Before using the functions listed below, you have to cancel M120 and the radius compensation:
 - Cycle **32** Tolerance
 - Cycle **19** Working plane
 - PLANE function
 - M114
 - M128
 - TCPM FUNCTION

Programming: Miscellaneous functions

10.4 Miscellaneous functions for path behavior

Superimposing handwheel positioning during program run: M118

Standard behavior

In the program run modes, the TNC moves the tool as defined in the part program.

Behavior with M118

M118 permits manual corrections by handwheel during program run. Just program M118 and enter an axis-specific value (linear or rotary axis) in millimeters.



The handwheel superimpositioning function with M118 in combination with collision monitoring is only possible in stopped condition. To be able to use M118 without limitations, you have to deselect DCM either by soft key in the menu, or activate a kinematics model without collision monitored objects (CMOs)

Input

If you enter M118 in a positioning block, the TNC continues the dialog for this block by asking you the axis-specific values. The coordinates are entered with the orange axis direction buttons or the ASCII keyboard.

Effect

Cancel handwheel positioning by programming M118 once again without coordinate input.

M118 becomes effective at the start of block.

Example NC blocks

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 $\pm 5^\circ$ from the programmed value:

```
L X+0 Y+38.5 RL F125 M118 X1 Y1 B5
```



M118 is effective in a tilted coordinate system if you activate the tilted working plane function for the Manual Operation mode. If the tilted working plane function is not active for the Manual Operation mode, the original coordinate system is effective. M118 also functions in the Positioning with MDI mode of operation!

Virtual tool axis VT

Your machine tool builder must have prepared the TNC for this function. Refer to your machine manual.

With the virtual tool axis you can also traverse in the direction of a sloping tool with the handwheel with machines with swivel heads. To traverse in a virtual tool axis direction select the VT axis on the display of your handwheel, see "Traverse with electronic handwheels", page 520. With an HR 5xx handwheel you can select the virtual axis directly with the orange VI axis key if required (refer to your machine manual).

You can also carry out handwheel superimpositioning in the currently active tool axis direction with the M118 function. For this purpose, you must at least define the spindle axis with the permitted traverse range (e.g. M118 Z5) in the M118 function and select the VT axis on the handwheel.

Programming: Miscellaneous functions

10.4 Miscellaneous functions for path behavior

Retraction from the contour in the tool-axis direction: M140

Standard behavior

In the program run modes Program run single block and Program run full sequence the TNC moves the tool as defined in the part program.

Behavior with M140

With M140 MB (move back) you can enter a path in the direction of the tool axis for departure from the contour.

Input

If you enter M140 in a positioning block, the TNC continues the dialog and asks for the desired path of tool departure from the contour. Enter the requested path that the tool should follow when departing 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 TNC moves the tool along the entered path at rapid traverse.

Effect

M140 is effective only in the block in which it is programmed.

M140 becomes effective at the start of block.

Example NC blocks

Block 250: Retract the tool 50 mm from the contour.

Block 251: Move the tool to the limit of the traverse range.

```
250 L X+0 Y+38.5 F125 M140 MB 50 F750
```

```
251 L X+0 Y+38.5 F125 M140 MB MAX
```



M140 is also effective if the tilted-working-plane function is active. On machines with swivel heads, the TNC 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 a tool axis before entering **M140**, otherwise the direction of traverse is not defined.



Danger of collision!

If you modify the position of a rotary axis with the handwheel superimposition **M118** function and then run **M140**, the TNC ignores the superimposed values with the retraction movement.

This may cause undesired motion or collisions on machines with rotary axes in the head.

**Danger of collision!**

With **M140 MB MAX** and collision monitoring simultaneously possible (option #40 enabled, kinematics with defined collision objects selected), tool retraction does not terminate at the border of the traverse range but before a pending collision. Detected collisions with this constellation do not generate an error message, but the TNC continues the NC program from this position. This may cause unexpected movements or collisions with the workpiece.

Only use numerical values with collision monitoring **M140**.

Check in the **Program run, single block** operating mode whether the actual distance enables collision-free continuation of the NC program.

If you require **M140 MB MAX**, you must select kinematics without defined collision objects. Please note that with kinematics without defined collision objects, collision monitoring is **not** possible.

Programming: Miscellaneous functions

10.4 Miscellaneous functions for path behavior

Suppressing touch probe monitoring: M141

Standard behavior

When the stylus is deflected, the TNC outputs an error message as soon as you attempt to move a machine axis.

Behavior with M141

The TNC 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.



Danger of collision!

If you use M141, make sure that you retract the touch probe in the correct direction.

M141 functions only for movements with straight-line blocks.

Effect

M141 is effective only in the block in which it is programmed.

M141 becomes effective at the start of 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 TNC erases a programmed basic rotation from the NC program.



The function **M143** is not permitted during mid-program startup.

Effect

M143 is effective only in the block in which it is programmed.
M143 becomes effective at the start of the block.

Programming: Miscellaneous functions

10.4 Miscellaneous functions for path behavior

Automatically retract tool from the contour at an NC stop: M148

Standard behavior

At an NC stop the TNC stops all traverse movements. The tool stops moving at the point of interruption.

Behavior with M148



The M148 function must be enabled by the machine tool builder. The machine tool builder defines in a machine parameter the path that the TNC is to traverse for a **LIFTOFF** command.

The TNC retracts the tool by up to 2 mm in the direction of the tool axis if, in the **LIFTOFF** column of the tool table, you set the parameter **Y** for the active tool, see "Enter tool data into the table", page 174.

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



Danger of collision!

Remember that, especially on curved surfaces, the surface can be damaged during return to the contour. Retract the tool before returning to the contour!

In the **CfgLiftOff** machine parameter, define the value by which the tool is to be retracted. In the **CfgLiftOff** machine parameter you can also switch the function off.

Effect

M148 remains in effect until deactivated with M149.

M148 becomes effective at the start of block, M149 at the end of block.

Rounding corners: M197

Standard behavior

The TNC inserts a transition arc at outside corners with active radius compensation. This may lead to grinding of the edge.

Behavior with M197

With Function M197 the contour at the corner is tangentially extended and a smaller transition arc is then inserted. When you program Function M197 and then press the ENT key, the TNC opens the **DL** input field. In **DL** you define the length with which the TNC extends the contour elements. With M197 the corner radius is reduced, the corner grinds less and the traverse movement is still tangential.

Effect

The Function M197 is effective blockwise and is only effective on outside corners.

Example NC blocks

```
L X... Y... RL M197 DL0.876
```


11

**Programming:
Special functions**

Programming: Special functions

11.1 Overview of special functions

11.1 Overview of special functions

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

Function	Description
Dynamic Collision Monitoring with integrated fixture management (Option #40)	page 393
Adaptive Feed Control AFC (Option #45)	page 400
Active Chatter Control (Option #145)	page 412
Working with text files	page 423
Working with freely definable tables	page 427

Press the **SPEC FCT** and the corresponding soft keys to access further special functions of the TNC. The following tables will give you an overview of which functions are available.

Main menu for SPEC FCT special functions



► Press the special functions key

Soft key	Function	Description
	Define program defaults	page 391
	Functions for contour and point machining	page 391
	Define the PLANE function	page 439
	Define different conversational functions	page 392
	Define turning functions	page 489
	Define structure items	page 145

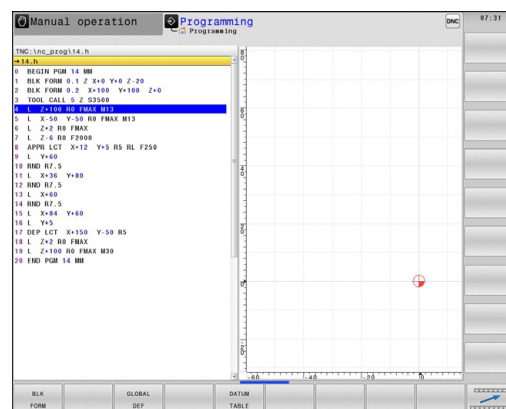


After pressing the SPEC FCT key, you can open the **smartSelect** selection window with the GOTO key. The TNC displays a structure overview with all available functions. You can rapidly navigate with the cursor or mouse and select functions in the tree diagram. The TNC displays online help for the specific functions in the window on the right.

PROGRAM DEFAULTS

- Select the program defaults menu

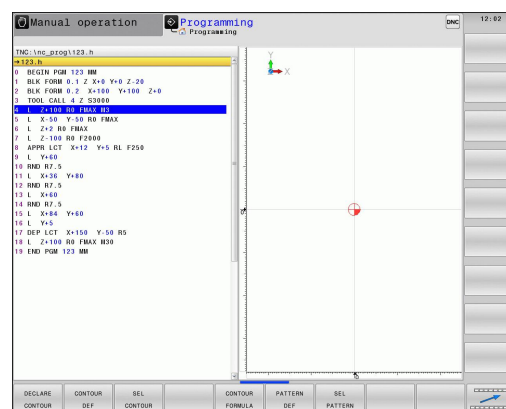
Soft key	Function	Description
BLK FORM	Define workpiece blank	page 102
DATUM TABLE	Select datum table	See User's Manual for Cycles
GLOBAL DEF	Define global cycle parameters	See User's Manual for Cycles



**CONTOUR
+ POINT
MACHINING**

- Select the menu for functions for contour and point machining

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



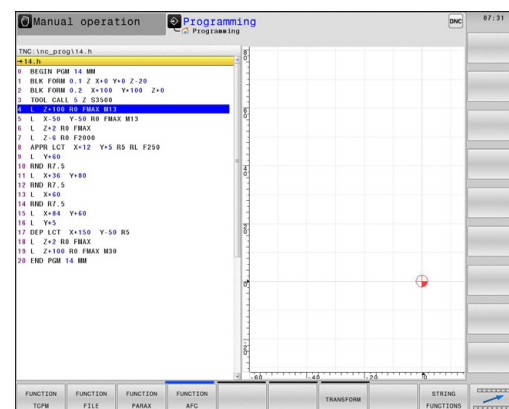
11

Programming: Special functions

11.1 Overview of special functions

Menu of various conversational functions

PROGRAM FUNCTIONS	► Select the menu for defining various conversational functions	
Soft key	Function	Description
FUNCTION TCPM	Define the positioning behavior for rotary axes	page 469
FUNCTION FILE	Define file functions	page 419
FUNCTION PARAX	Define the positioning behavior for parallel axes U, V, W	page 414
FUNCTION AFC	Define Adaptive Feed Control	page 400
TRANSFORM	Define coordinate transformations	page 420
STRING FUNCTIONS	Define string functions	page 345
FUNCTION FEED	Define dwell time	page 433
FUNCTION DCM	Define Dynamic Collision Monitoring DCM	page 393
INSERT COMMENT	Add comments	page 142



11.2 Dynamic Collision Monitoring (Option #40)

Function

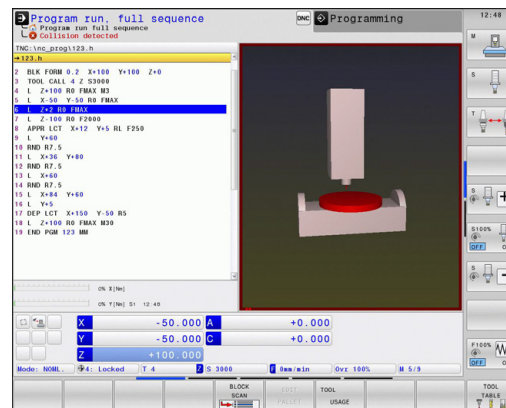


The Dynamic Collision Monitoring (DCM) must be adapted by the machine manufacturer for the TNC and for the machine. Refer to your machine manual.

The machine manufacturer can define any objects that are monitored by the TNC during all machining operations. If two objects monitored for collision come within a defined distance of each other, the TNC outputs an error message and terminates the movement.

The TNC can display the defined collision objects graphically in all machine modes of operation, see "Graphic display of the collision objects", page 394.

The TNC also monitors the active tool for collision and graphically displays it accordingly. The TNC always assumes cylindrical tools, and only considers lengths, radii and corresponding oversizes entered in the tool table. The TNC likewise monitors stepped tools according to the definition in the tool table.



Generally valid constraints:

- DCM helps to reduce the danger of collision. However, the TNC cannot consider all possible constellations in operation.
- Collisions between machine components and the tool and between the tool and the workpiece are not detected by the TNC.
- DCM can only protect those machine components from collision that your machine tool builder has correctly defined with regard to dimensions, orientation and position.
- The TNC 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 TNC no longer monitors the stylus length and ball-tip diameter so that you can also probe in 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.
- The tool oversizes **DL** and **DR** from the tool table are taken into account by the TNC. Tool oversizes from the **TOOL CALL** block are not taken into account.

Programming: Special functions

11.2 Dynamic Collision Monitoring (Option #40)

Graphic display of the collision objects

Activate the graphic display of the collision objects as follows:

- Select any machine mode of operation



- Press the screen switchover key

POSITION
+
KINEMATICS

- Select the desired screen layout

PROGRAM
+
KINEMATICS

KINEMATICS

You can also use the soft keys to change the display of the collision objects.





Modify the graphic display of the collision objects as follows:

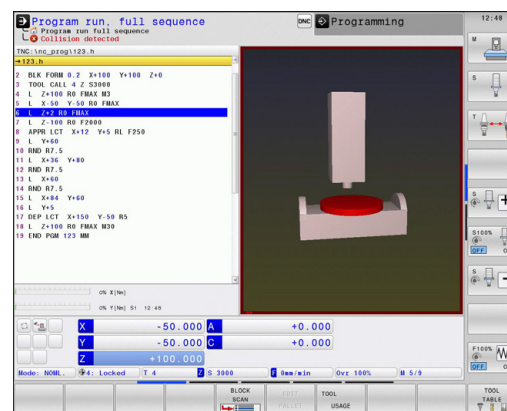
- Shift the soft-key row if necessary

KINEMATICS

- Press the **KINEMATICS** soft key
- Modify the graphic display of the collision objects with the following functions

The following functions are available:

Soft key	Function
	Switch between wire-frame and solid-object view
	Switch between shadowed and transparent view
	Display/hide the coordinate systems that result from transformations in the kinematics description
	Functions for rotating, zooming and shifting



You can also use the mouse to change the display of the collision objects.

The following functions are available:

- ▶ In order to rotate the model shown in three dimensions you hold the right mouse button down and move the mouse. If you simultaneously press the shift key, you can only rotate the model horizontally or vertically.
- ▶ To shift the model shown: Hold the center mouse button or the wheel button down and move the mouse. If you simultaneously press the shift key, you can only shift the model horizontally or vertically.
- ▶ To zoom in on a certain area: Mark a zoom area by holding the left mouse button down. After you release the left mouse button, the TNC zooms in on the defined area.
- ▶ To rapidly magnify or reduce any area: Rotate the mouse wheel backwards or forwards.
- ▶ To return to the standard display: Press the shift key and simultaneously double-click with the right mouse key. The rotation angle is maintained if you only double-click with the right mouse key.

Programming: Special functions

11.2 Dynamic Collision Monitoring (Option #40)

Collision monitoring in the manual operating modes

In the **Manual operation** and **Electronic handwheel** operating modes, the TNC stops a motion if two objects monitored for collision approach each other within a distance of less than 2 mm. In this case, the TNC displays an error message naming the two objects causing collision.

Before the collision warning, the TNC dynamically reduces the feed of the movements to ensure that the axes stop in good time before a collision.

If you have selected a screen layout in which collision objects are on the right, then the TNC additionally marks the colliding objects in red.



Once a collision warning is displayed, machine motions via the direction keys or handwheel are possible only if the motion increases the distance between the collision objects.

Motions that reduce the distance or leave it unchanged are not allowed as long as collision monitoring is active.

To deactivate collision monitoring, see "Activating and deactivating collision monitoring", page 398.



Note the generally valid constraints, see "Function", page 393

Collision monitoring in the Program Run operating modes

In the **Positioning with manl.data input**, **Program run single block** and **Program run, full sequence** operating modes, the TNC stops the program run before the block in which two objects monitored for collision would approach each other within a distance of less than 5 mm. In this case, the TNC displays an error message naming the two objects causing collision.

If you have selected a screen layout in which the collision objects are on the right, then the TNC additionally marks the colliding objects in red.



The TNC monitors motions blockwise. The TNC interrupts the program run with a collision warning in the block that would cause a collision.



Danger of collision!

With **M140 MB MAX** and collision monitoring simultaneously possible (option #40 enabled, kinematics with defined collision objects selected), tool retraction does not terminate at the border of the traverse range but before a pending collision. Detected collisions with this constellation do not generate an error message, but the TNC continues the NC program from this position. This may cause unexpected movements or collisions with the workpiece.

Only use numerical values with collision monitoring **M140**.

Check in the **Program run, single block** operating mode whether the actual distance enables collision-free continuation of the NC program.

If you require **M140 MB MAX**, you must select kinematics without defined collision objects. Please note that with kinematics without defined collision objects, collision monitoring is **not** possible.



Constraints with program run:

- For tapping with a floating tap holder only the basic setting of the floating tap holder is taken into account with collision monitoring.
- The handwheel superimpositioning function with **M118** in combination with active collision monitoring is only possible in stopped condition.
- The TNC is not able to implement collision monitoring if functions or cycles require the coupling of several axes e.g. with eccentric turning.
- The TNC is not able to implement collision monitoring if at least one axis is referenced in lag tracking or not.

Also note the generally valid constraints, see "Function", page 393

11.2 Dynamic Collision Monitoring (Option #40)

Activating and deactivating collision monitoring

In some cases it is necessary to temporarily deactivate collision monitoring:

- To reduce the distance between two objects monitored for collision
- To prevent stops during program runs



Danger of collision!

If you deactivate collision monitoring, the TNC does not output an error message with a pending collision.

Also, with inactive collision monitoring, the TNC does not prevent movements caused by collision.

Permanently manually activating and deactivating collision monitoring



- ▶ Select the **Manual operation** or **Electronic handwheel** mode of operation



- ▶ Shift the soft-key row if necessary



- ▶ Press the **COLLISION** soft key



- ▶ Select the operating modes for which the modification should apply:
 - **Program run: Positioning with manl.data input, Program run, single block and Program run, full sequence**
 - **Manual Operation: Manual operation and Electronic handwheel**



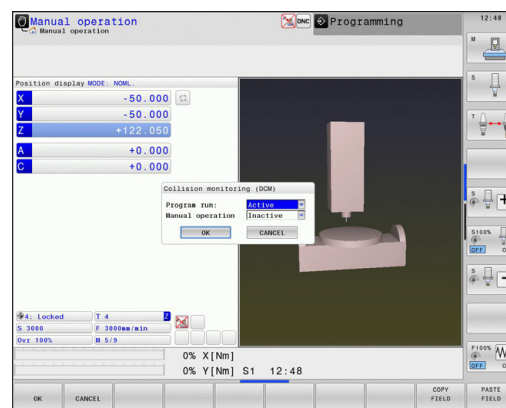
- ▶ Press the **GO TO** key



- ▶ Select the condition for which the selected operating modes should apply:
 - **Inactive:** Deactivate collision monitoring
 - **Active:** Activate collision monitoring



- ▶ Press the **OK** soft key.



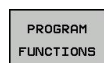
Dynamic Collision Monitoring (Option #40) 11.2

Temporarily activating and deactivating collision monitoring via program control

- ▶ Open the NC program in the **Programming** mode of operation
- ▶ Place the cursor at the desired position, e.g. before Cycle 800 to enable eccentric turning



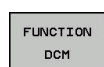
- ▶ Press the **SPEC FCT** key



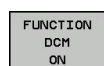
- ▶ Press the **PROGRAM FUNCTIONS** soft key



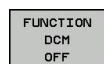
- ▶ Shift the soft-key row



- ▶ Press the **FUNCTION DCM** soft key.



- ▶ Select the condition with the corresponding soft key:



- **FUNCTION DCM ON:** Activate collision monitoring
- **FUNCTION DCM OFF:** Deactivate collision monitoring

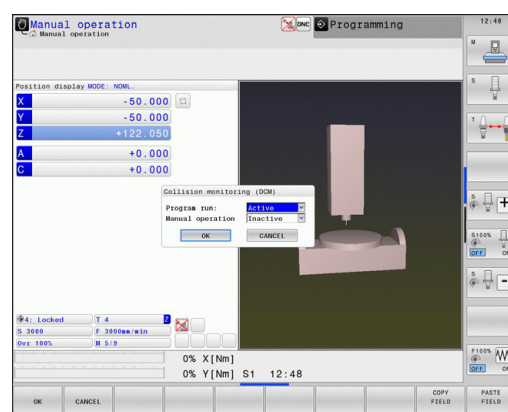


The settings applied with the **FUNCTION DCM** function are only effective in the active NC program. After terminating the program run or selecting a new program, the settings made for **Program Run** and **Manual Operation** with the soft key again become effective: see "Permanently manually activating and deactivating collision monitoring", page 398

Symbols

Symbols in the status display show the condition of collision monitoring:

Icon	Function
	Collision monitoring active
	Collision monitoring is not available
	Collision monitoring is not active



11.3 Adaptive feed control AFC (Option #45)

11.3 Adaptive feed control AFC (Option #45)

Application



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

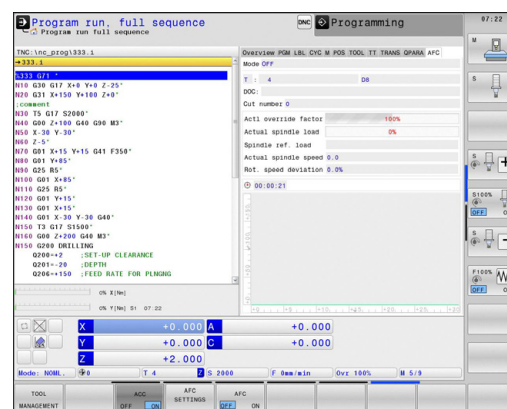
Refer to your machine manual.

Your machine tool builder may also have specified whether the TNC uses the spindle power or any other value as the input value for the feed control.



Adaptive feed control is not intended for tools with diameters less than 5 mm. This limit diameter might also be greater if the spindle's rated power is very high.

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 TNC automatically controls the feed rate during program run as a function of the current spindle power consumption. The spindle power required for each machining step is to be recorded in a teach-in cut and saved by the TNC in a file belonging to the part program. When each machining step is started, which is normally when the spindle is switched on, the TNC controls the feed rate so that it remains within the limits that you have defined.

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) offers the following benefits:

- Optimization of machining time

By controlling the feed rate, the TNC tries to maintain the recorded maximum spindle power 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 maximum value, the TNC decreases the feed rate until the reference spindle power is reattained. If the maximum spindle power is exceeded during machining and at the same time the feed rate falls below the minimum that you defined, the TNC 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

Programming: Special functions

11.3 Adaptive feed control AFC (Option #45)

Defining the AFC basic settings

You make the control settings for the TNC feed rate control in the table **AFC.TAB**, which must be saved in the root directory **TNC:\table**.

The data in this table are default values that were copied during a teach-in cut into a file belonging to the respective program and serve as the basis for control. The following data are to be defined in this 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 TNC 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 TNC 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 (feed rate in the air). 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	<p>Desired reaction of the TNC to overload:</p> <ul style="list-style-type: none"> ■ 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 ■ -: No overload reaction <p>The TNC conducts a shutdown response if 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 ASCII keyboard.</p>

Column	Function
POUT	Spindle power at which the TNC is to detect tool exit from 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 TNC 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.



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 TNC uses permanently defined internal control settings for the teach-in cut. It is best, however, to work with the AFC.TAB table.

Proceed as follows to create the AFC.TAB file (only necessary if the file does not yet exist):

- ▶ Select the **Programming** mode of operation
- ▶ To call the file manager, press the **PGM MGT** key
- ▶ Select the **TNC:** directory
- ▶ Make the new file **AFC.TAB** and confirm with the **ENT** key: The TNC shows a list of table formats
- ▶ Select the **AFC.TAB** table format and confirm with the **ENT** key: The TNC creates a table with the **Standard** control settings

Programming: Special functions

11.3 Adaptive feed control AFC (Option #45)

Recording a teach-in cut

The TNC provides several cycles that enable you to start and stop a teach-in step:

- **FUNCTION AFC CTRL:** The AFC CTRL function activates closed-loop mode starting with the place at which this block is run (even if the teach-in phase has not yet been completed)
- **FUNCTION AFC CUT BEGIN TIME1 DIST2 LOAD3:** The TNC starts a sequence of cuts with active AFC. The switch from the teach-in cut to closed-loop mode begins as soon as the reference load was determined in the teach-in phase, or once one of the conditions TIME, DIST or LOAD is fulfilled. With TIME you define the maximum duration in seconds of the teach-in phase. DIST defines the maximum distance for the teach-in cut. With LOAD you can set a reference load directly.
- **FUNCTION AFC CUT END:** The AFC CUT END function deactivates the AFC control



The defaults TIME, DIST and LOAD are modally effective. They can be reset with the entry 0.

Programming AFC

To program the AFC functions for starting and ending the teach in cut, proceed as follows:

- ▶ In the **Programming** mode, press the SPEC FCT key
- ▶ Select the **PROGRAM FUNCTIONS** soft key
- ▶ Select the **FUNCTION AFC** soft key
- ▶ Select the function

With a teach-in cut the TNC firstly copies the basic settings defined in the AFC.TAB table into the file **<name>.H.AFC.DEP** for each machining step. **<name>** is the name of the NC program for which you have recorded the teach-in cut. In addition, the TNC measures the maximum spindle power consumed during the teach-in cut and saves this value in the table.

Each line in the **<name>.H.AFC.DEP** file stands for a machining section, that you start with **FUNCTION AFC CUT BEGIN** and complete with **FUNCTION AFC CUT END**. You can edit all data of the **<name>.H.AFC.DEP** file if you wish to optimize them. If you have optimized the values in comparison with the values in the AFC.TAB table, the TNC places an asterisk * in front of the control settings in the AFC column. Besides the data from the AFC.TAB table see "Defining the AFC basic settings", page 402, the TNC saves the following additional information in the **<name>.H.AFC.DEP** file:

Column	Function
NR	Number of the machining step
TOOL	Number or name of the tool with which the machining step was made (not editable)
IDX	Index of the tool with which the machining step was made (not editable)
N	Difference for tool call: <ul style="list-style-type: none"> ■ 0: Tool was called by its tool number ■ 1: Tool was called by its tool name
PREF	Reference load of the spindle. The TNC measures the value in percent with respect to the rated power of the spindle
ST	Status of the machining step: <ul style="list-style-type: none"> ■ L: In the next program run, a teach-in cut is recorded for this machining step. The TNC overwrites any existing values in this line ■ C: The teach-in cut was successfully completed. The next program run can be conducted with automatic feed control
AFC	Name of the control setting

Programming: Special functions

11.3 Adaptive feed control AFC (Option #45)

Remember the following before you record a teach-in cut:

- If required, adapt the control settings in the AFC.TAB table
- Enter the desired control setting for all tools in the **AFC** column of the tool table TOOL.T
- Select the program for teach-in
- Activate the adaptive feed control by soft key, see "Activating/deactivating AFC", page 407



You can teach any number of machining steps for a tool. Your machine tool builder will either make a function available for this, or will integrate this possibility in the functions for switching on the spindle. Refer to your machine manual.

The functions for starting and ending a machining step are machine-dependent. Refer to your machine manual.



When you are performing a teach-in cut, the TNC shows the spindle reference power determined until this time in a pop-up window.

You can reset the reference power at any time by pressing the **PREF RESET** soft key. The TNC then restarts the learning phase.

When you record a teach-in cut, the TNC internally sets the spindle override to 100 %. Then you can no longer change the spindle speed.

During the teach-in cut, you can influence the measured reference load by using the feed rate override to make any changes to the contouring feed rate.




You do not have to run the entire machining step in the learning mode. If the cutting conditions do not change significantly, you can switch to the control mode immediately. Press the **EXIT LEARNING** soft key, and the status changes from **L** to **C**.

You can repeat a teach-in cut as often as desired. Manually change the status from **ST** back to **L**. It may be necessary to repeat the teach-in cut if the programmed feed rate is far too fast, and forces you to sharply decrease the feed rate override during the machining step.

The TNC changes the status from teach-in (**L**) to controlling (**C**) only when the recorded reference load is greater than 2 %. Adaptive feed control is not possible for smaller values.

Adaptive feed control AFC (Option #45) 11.3

Proceed as follows to select and, if required, edit the **<name>.H.AFC.DEP** file:

-  ▶ Select the **Program Run, Full Sequence** operating mode
-  ▶ Shift the soft-key row
-  ▶ Select the table of AFC settings
- ▶ Make optimizations if required



Note that the **<name>.H.AFC.DEP** file is locked against editing as long as the NC program **<name>.H** is running.

The TNC removes the editing lock if one of the following functions has been executed:





- **M02**
- **M30**
- **END PGM**

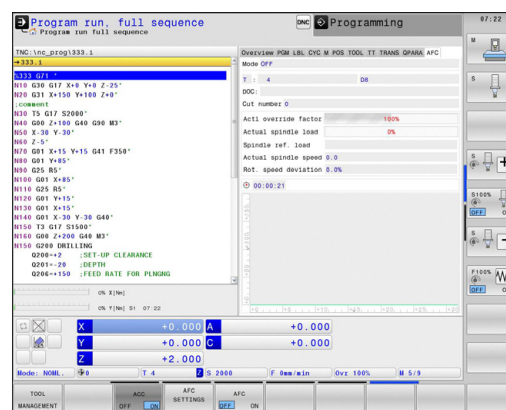
You can also change the **<name>.H.AFC.DEP** file in the **Programming** mode of operation. If necessary, you can even delete a machining step (entire line) there.



In order to edit the **<name>.H.AFC.DEP** file, you must first set the file manager so that all file types can be displayed (**SELECT TYPE** soft key) Also see: "Files", page 114

Activating/deactivating AFC

-  ▶ Select the **Program Run, Full Sequence** operating mode
-  ▶ Shift the soft-key row
-  ▶ To activate the adaptive feed control: Set the soft key to **ON**, and the TNC displays the AFC symbol in the position display, see "Status displays", page 78
-  ▶ To deactivate the adaptive feed control: Set the soft key to **OFF**



11.3 Adaptive feed control AFC (Option #45)



The adaptive feed control remains active until you deactivate it by soft key. The TNC remembers the setting of the soft key even if the power is interrupted.

If the adaptive feed control is active in the **control** mode, the TNC internally sets the spindle override to 100 %. Then you can no longer change the spindle speed.


If the adaptive feed control is active in the **control** mode, the TNC takes over the feed rate override function:

- If you increase the feed rate override, it has no influence on the control.
- If you decrease the feed rate override by more than **10%** with respect to the maximum setting, the TNC switches the adaptive feed control off. In this case the TNC displays a window to inform you.

In NC blocks containing **FMAX**, the adaptive feed control is **not active**.

Mid-program startup is allowed during active feed control and the TNC takes the cut number of the startup point into account.

In the additional status display, the TNC displays various information when the adaptive feed control is active, see "Additional status displays", page 79.

In addition, the TNC shows the symbol  in the position display.

Log file

The TNC stores various pieces of information for each machining step of a teach-in cut in the **<name>.H.AFC2.DEP** file. **<name>** is the name of the NC program for which you have recorded the teach-in cut. During control, the TNC updates the data and makes various evaluations. The following data are to be saved in this table:

Column	Function
NR	Number of the machining step
TOOL	Number or name of the tool with which the machining step was made
IDX	Index of the tool with which the machining step was made
SNOM	Nominal spindle speed [rpm]
SDIF	Maximum difference of the spindle speed in % of the nominal speed
LTIME	Machining time for the teach-in cut
CTIME	Machining time for the control cut
TDIFF	Time difference in % between the machining time during teach-in and control
PMAX	Maximum recorded spindle power during machining. The TNC shows the value as a percent of the spindle's rated power.
PREF	Reference load of the spindle. The TNC shows the value as a percent of the spindle's rated power.
FMIN	Smallest occurring feed factor. The TNC shows the value as a percentage of the programmed feed rate
OVLD	Reaction by the TNC to overload: <ul style="list-style-type: none"> ■ M: A macro defined by the machine tool builder has been run ■ S: Immediate NC stop was conducted ■ F: NC stop was conducted after the tool was retracted ■ E: An error message was displayed ■ -: There was no overload reaction
BLOCK	Block number at which the machining step begins

Programming: Special functions

11.3 Adaptive feed control AFC (Option #45)



The TNC records the total machining time for all teach-in cuts (**LTIME**), all control cuts (**CTIME**) and the total time difference (**TDIFF**), and enters it after the keyword **TOTAL** in the last line of the log file.

The TNC can only calculate the time difference (**TDIFF**) if you have completed the teach-in step. Otherwise the column remains empty.

Proceed as follows to select the **<name>.H.AFC2.DEP** file:



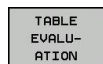
- ▶ Select the **Program Run, Full Sequence** operating mode



- ▶ Shift the soft-key row



- ▶ Select the table of AFC settings



- ▶ Show the log file

Tool breakage/tool wear monitoring



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

Refer to your machine manual.

With the breakage/wear monitor, a cut-based tool breakage detection during active AFC can be realized.

Through the functions that can be defined by the machine tool builder you can define a percentage value for wear or breakage detection with respect to the rated power.

When the defined limit spindle power range is not maintained, the TNC conducts an NC stop.

Spindle load monitoring



This feature must be enabled and adapted by the machine tool builder. Refer to your machine manual.

With the spindle load monitoring function you can easily have the spindle load monitored in order, for example, to detect overloading the spindle power.

The function is independent of AFC, i.e. it is not cut-based and does not depend on teach-in steps. Through the functions that can be defined by the machine tool builder, you only need to define the percentage value for spindle limit power with respect to the rated power.

When the defined limit spindle power range is not maintained, the TNC conducts an NC stop.

Programming: Special functions

11.4 Active Chatter Control ACC (Option #145)

11.4 Active Chatter Control ACC (Option #145)

Application



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

Refer to your machine manual.

Strong milling forces come into play during roughing (power milling). Depending on the tool spindle speed, the resonances in the machine tool and the chip volume (metal-removal rate during milling), the tool can sometimes begin to “chatter.” This chattering places heavy strain on the machine, and causes ugly marks on the workpiece surface. The tool, too, is subject to heavy and irregular wear from chattering. In extreme cases it can result in tool breakage.

To reduce the inclination to chattering, HEIDENHAIN now offers an effective antidote with **ACC (Active Chatter Control)**. The use of this control function is particularly advantageous during heavy cutting. ACC makes substantially higher metal removal rates possible. This makes it possible to increase your metal removal rate by up to 25 % and more, depending on the type of machine. You reduce the mechanical load on the machine and increase the life of your tools at the same time.



Please note that ACC was developed especially for heavy cutting and is particularly effective in this area. You need to conduct appropriate tests to ensure whether ACC is also advantageous during standard roughing.

When you use the ACC feature, you must enter the number of tool cuts **CUT** for the corresponding tool in the TOOL.T tool table.

Activating/deactivating ACC

To activate ACC, for the respective tool in the tool table TOOL.T you first need to set the column **ACC** to **Y** (ENT key = Y, NO ENT key = N).

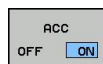
To activate; deactivate ACC for the machine mode,



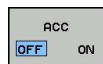
- ▶ Select the **Program Run, Full Sequence**, the **Program Run, Single Block** or the **Positioning with Manual Data Input** mode of operation



- ▶ Shift the soft-key row



- ▶ Activate ACC: Set the soft key to **ON** and the TNC displays the ACC symbol in the position display, see "Status displays", page 78



- ▶ To deactivate ACC: Set the soft key to **OFF**

If ACC is on, in the position display the TNC shows the symbol **ACC**.

11.5 Working with the Parallel Axes U, V and W

11.5 Working with the Parallel Axes U, V and W

Overview



Your machine must be configured by the machine manufacturer if you want to use parallel-axis functions.

Depending on the configuration, the PARAXCOMP function may be activated as standard.

Refer to your machine manual.

The axes U, V and W are secondary axes parallel to the principal axes X, Y and Z, respectively. Principal axes and parallel axes are permanently assigned to each other.

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

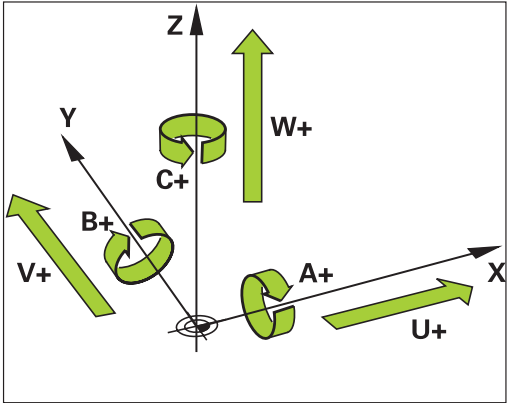
The TNC provides the following functions for machining with the parallel axes U, V and W:

Soft key	Function	Meaning	Page
FUNCTION PARAXCOMP	PARAXCOMP	Define the TNC's behavior when positioning parallel axes	416
FUNCTION PARAXMODE	PARAXMODE	Define the axes the TNC is to use for machining	416



After the TNC is started up, the standard configuration is always effective.






You must deactivate the parallel-axis functions before switching the machine kinematics.



FUNCTION PARAXCOMP DISPLAY

Use the **PARAXCOMP DISPLAY** function to activate the display function for parallel axis movements. The TNC considers the traverse movements of the parallel axis in the position display of the associated principal axis (sum display). Therefore, the position display of the principal axis always displays the relative distance from the tool to the workpiece, regardless of whether you move the principal axis or the minor axis.

Proceed as follows for the definition:

-  ► Show the soft-key row with special functions
-  ► Select the menu for defining various plain-language functions
-  ► Select **FUNCTION PARAX**
-  ► Select **FUNCTION PARAXCOMP**
-  ► Select **FUNCTION PARAXCOMP DISPLAY**
- Define the parallel axis whose movements the TNC is to take into account in the position display of the associated principal axis

NC block

13 FUNCTION PARAXCOMP DISPLAY W

FUNCTION PARAXCOMP MOVE








The **PARAXCOMP MOVE** function can be used only in connection with straight-line blocks (**L**).

The TNC uses the **PARAXCOMP MOVE** function to compensate for the movement of a parallel axis by performing a compensation movement in the associated principal axis.

For example, if a parallel axis moves in the negative W-axis direction, the principal axis Z simultaneously moves in the positive direction by the same value. The relative distance from the tool to the workpiece remains the same. Application in gantry-type milling machine: Retract the spindle sleeve to move the cross beam down simultaneously.

Proceed as follows for the definition:

-  ► Show the soft-key row with special functions
-  ► Select the menu for defining various plain-language functions
-  ► Select **FUNCTION PARAX**
-  ► Select **FUNCTION PARAXCOMP**
-  ► Select **FUNCTION PARAXCOMP MOVE**
- Define the parallel axis

NC block

13 FUNCTION PARAXCOMP MOVE W

Programming: Special functions

11.5 Working with the Parallel Axes U, V and W

Deactivating FUNCTION PARAXCOMP



After the TNC is started up, the standard configuration is always effective.

The parallel-axis function **PARAXCOMP** is automatically reset by the TNC with the following functions:

- Selection of a program
- **PARAXCOMP OFF**

You must deactivate the parallel-axis functions before switching the machine kinematics.

Use the **PARAXCOMP OFF** function to switch off the parallel-axis functions **PARAXCOMP DISPLAY** and **PARAXCOMP MOVE**. Proceed as follows for the definition:

SPEC
FCT

- ▶ Show the soft-key row with special functions

PROGRAM
FUNCTIONS

- ▶ Select the menu for defining various plain-language functions

FUNCTION
PARAX

- ▶ Select **FUNCTION PARAX**

FUNCTION
PARAXCOMP

- ▶ Select **FUNCTION PARAXCOMP**

FUNCTION
PARAXCOMP
OFF

- ▶ Select **FUNCTION PARAXCOMP OFF**. If you want to switch off the parallel-axis functions only for individual parallel axes, then the respective axis must be specifically indicated.

FUNCTION PARAXMODE



To activate the **PARAXMODE** function, you must always define three axes.

If you combine the **PARAXMODE** and **PARAXCOMP** functions, the TNC deactivates the **PARAXCOMP** function for an axis that was defined in both functions. When you deactivate **PARAXMODE**, the **PARAXCOMP** function becomes active again.

Use the **PARAXMODE** function to define the axes the TNC is to use for machining. You program all traverse movements and contour descriptions in the principal axes X, Y and Z, independent of your machine.

Define the three axes in the **PARAXMODE** function (e.g. **FUNCTION PARAXMODE X Y W**), which the TNC is to use to execute the programmed traverse movements.

Proceed as follows for the definition:

SPEC
FCT

- ▶ Show the soft-key row with special functions

PROGRAM
FUNCTIONS

- ▶ Select the menu for defining various plain-language functions

NC blocks

13 FUNCTION PARAXCOMP OFF

13 FUNCTION PARAXCOMP OFF W

NC block

13 FUNCTION PARAXMODE X Y W

Working with the Parallel Axes U, V and W 11.5

FUNCTION
PARAX

- ▶ Select **FUNCTION PARAX**

FUNCTION
PARAXMODE

- ▶ Select **FUNCTION PARAXMODE**

FUNCTION
PARAXMODE

- ▶ Select **FUNCTION PARAXMODE**
- ▶ Define the axes for machining

Move the principal axis and the parallel axis simultaneously

If the **PARAXMODE** function is active, the TNC uses the axes defined in the function to execute the programmed traverse movements. If the TNC is to traverse a parallel axis simultaneously with the associated principal axis, you can identify the respective axis by additionally entering the character "&". The axis with the & character then refers to the principal axis.



The syntax element "&" is only permitted in L blocks. Additional positioning of a principal axis with the "&" command is done in the REF system. If you have set the position display to "actual value", this movement will not be shown. If necessary, switch the position display to "REF value".

NC block

```
13 FUNCTION PARAXMODE X Y W
```

```
14 L Z+100 &Z+150 R0 FMAX
```

Deactivating FUNCTION PARAXMODE



After the TNC is started up, the standard configuration is always effective.

The parallel-axis function **PARAXMODE OFF** is automatically reset by the TNC via the following functions:

- Selection of a program
- End of program
- M2 or M30
- **PARAXMODE OFF**

You must deactivate the parallel-axis functions before switching the machine kinematics.

NC block

```
13 FUNCTION PARAXMODE OFF
```

Use the **PARAXMODE OFF** function to switch off the parallel-axis function. The TNC then uses the principal axes defined by the machine manufacturer. Proceed as follows for the definition:

SPEC
FCT

- ▶ Show the soft-key row with special functions

PROGRAM
FUNCTIONS

- ▶ Select the menu for defining various plain-language functions

FUNCTION
PARAX

- ▶ Select **FUNCTION PARAX**

FUNCTION
PARAXMODE

- ▶ Select **FUNCTION PARAXMODE**

FUNCTION
PARAXMODE
OFF

- ▶ Select **FUNCTION PARAXMODE OFF**

Programming: Special functions

11.5 Working with the Parallel Axes U, V and W

Example: Drilling in the W axis

0 BEGIN PGM PAR MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 5 Z S2222	Call the tool in the spindle axis Z
4 L Z+0 W+0 R0 FMAX M91	Reset the principle axis and minor axis
5 L Z+100 R0 FMAX M3	Position the principal axis
6 CYCL DEF 200 DRILLING	
Q200=+2 ;SET-UP CLEARANCE	
Q201=-20 ;DEPTH	
Q206=+150 ;FEED RATE FOR PLNGNG	
Q202=+5 ;	
Q210=+0 ;DWELL TIME AT TOP	
Q203=+0 ;SURFACE COORDINATE	
Q204=+50 ;2ND SET-UP CLEARANCE	
Q211=+0 ;DWELL TIME AT DEPTH	
Q395=+0 ;DEPTH REFERENCE	
7 FUNCTION PARAXCOMP DISPLAY Z W	Activate display compensation
8 FUNCTION PARAXMODE X Y W	Positive axis selection
9 L X+50 Y+50 R0 FMAX M99	Infeed runs minor axis W
10 FUNCTION PARAXMODE OFF	Restore standard axis configuration
11 L Z+0 W+0 R0 FMAX M91	Reset the principle axis and minor axis
12 L M30	
13 END PGM PAR MM	

11.6 File functions

Application

The **FILE FUNCTION** features are used to copy, move and delete files from within the part program.



You must not use **FILE** functions on programs or files, to which you have previously made reference with functions such as **CALL PGM** or **CYCL DEF 12 PGM CALL**.

Defining file functions

SPEC
FCT

- Press the special functions key

PROGRAM
FUNCTIONS

- Select the program functions

FUNCTION
FILE

- Select the file functions: The TNC displays the available functions

Soft key	Function	Meaning
FILE COPY	FILE COPY	Copy file: Enter the name and path of the file to be copied, as well as the target path
FILE MOVE	FILE MOVE	Move file: Enter the name and path of the file to be moved, as well as the target path
FILE DELETE	FILE DELETE	Delete file: Enter the path and name of the file to be deleted

Programming: Special functions

11.7 Definition of a datum shift






11.7 Definition of a datum shift

Overview

As an alternative to the coordinate transformation Cycle 7 **DATUM SHIFT**, you can use the **TRANS DATUM** plain-language function. Just as in Cycle 7, you can use **TRANS DATUM** to directly program shift values or activate a line from a selectable datum table. In addition, there is also the **TRANS DATUM RESET** function, which you can easily use to reset a datum shift.

TRANS DATUM AXIS

You can define a datum shift by entering values in the respective axes with the **TRANS DATUM AXIS** function. You can define up to nine coordinates in one block, and incremental entries are possible. Proceed as follows for the definition:

- 
 - ▶ Show the soft-key row with special functions
- 
 - ▶ Select the menu for defining various plain-language functions
- 
 - ▶ Select transformations
- 
 - ▶ Select datum shifting with **TRANS DATUM**
- 
 - ▶ Select the value input soft key
 - ▶ Enter the datum shift in the affected axes, confirming with the **ENT** key each time









Values entered as absolute values refer to the workpiece datum, which is specified either by datum setting or with a preset from the preset table. Incremental values always refer to the datum which was last valid (this may be a datum which has already been shifted).

NC block

```
13 TRANS DATUMAXIS X+10 Y+25 Z+42
```


TRANS DATUM TABLE

You can define a datum shift by selecting a datum number from a datum table with the **TRANS DATUM TABLE** function. Proceed as follows for the definition:

- 
 - ▶ Show the soft-key row with special functions
- 
 - ▶ Select the menu for defining various plain-language functions
- 
 - ▶ Select transformations
- 
 - ▶ Select datum shifting with **TRANS DATUM**
- 
 - ▶ Reset the cursor to the function **TRANS AXIS**
- 
 - ▶ Select datum shifting with **TRANS DATUM TABLE**
 - ▶ If desired, enter the name of the datum table from which you want to activate the datum number, and confirm with the **ENT** key. If you do not want to define a datum table, confirm with the **NO ENT** key
 - ▶ Enter the line number to be activated by the TNC, and confirm with the **ENT** key



If you did not define a datum table in the **TRANS DATUM TABLE** block, then the TNC uses the datum table already selected in the NC program with **SEL TABLE**, or the datum table with status M selected in the **Program Run, Single Block** or **Program Run, Full Sequence** operating mode.

NC block

13 TRANS DATUMTABLE TABLINE25

11.7 Definition of a datum shift

TRANS DATUM RESET

Use the **TRANS DATUM RESET** function to cancel a datum shift. How you previously defined the datum is irrelevant. Proceed as follows for the definition:

NC block

13 TRANS DATUM RESET

- SPEC
FCT

▶ Show the soft-key row with special functions
- PROGRAM
FUNCTIONS

▶ Select the menu for defining various plain-language functions
- TRANSFORM

▶ Select transformations
- TRANS
DATUM

▶ Select datum shifting with **TRANS DATUM**
- RESET
DATUM
SHIFT

▶ Select the RESET DATUM SHIFT soft key

11.8 Creating Text Files

Application

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






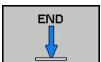
- Recording test results
- Documenting working procedures
- Creating formula collections

Text files are type .A files (ASCII files). If you want to edit other types of files, you must first convert them into type .A files.

Opening and exiting text files

- ▶ Select the **Programming** mode of operation
- ▶ To call the file manager, press the **PGM MGT** key.
- ▶ Display type .A files: Press the **SELECT TYPE** and then the **SHOW .A** soft keys
- ▶ 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 a part 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
	Go to beginning of file
	Go to end of file

11.8 Creating Text Files

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.

The line in which the cursor is presently located is depicted in a different color. 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.

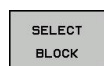
- ▶ Move the cursor to the word or line that you wish to erase and insert at a different place in the text
- ▶ Press the **DELETE WORD** or **DELETE LINE** soft key. The text is placed in the buffer memory
- ▶ Move the cursor to the location where you wish to insert the text, and press the **RESTORE LINE/WORD** soft key

Soft key	Function
<div>DELETE LINE</div>	Delete and temporarily store a line
<div>DELETE WORD</div>	Delete and temporarily store a word
<div>DELETE CHAR</div>	Delete and temporarily store a character
<div>INSERT LINE / WORD</div>	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.



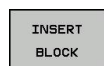
- ▶ Press the **SELECT BLOCK** soft key
- ▶ Move the cursor to the last character of the text you wish to select. You can select whole lines by moving the cursor up or down directly with the arrow keys—the selected text is shown in a different color

After selecting the desired text block, you can edit the text with the following soft keys:

Soft key	Function
	Delete the selected block and store temporarily
	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

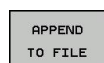


- ▶ Press the **INSERT BLOCK** soft key: The text block is inserted.

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



- ▶ Press the **APPEND TO FILE** soft key. The TNC displays the dialog prompt **Destination file =**
- ▶ Enter the path and name of the destination file. The TNC appends the selected text to the specified file. If no target file with the specified name is found, the TNC 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



- ▶ Press the **READ FILE** soft key. The TNC displays the dialog prompt **File name =**
- ▶ Enter the path and name of the file you want to insert

11.8 Creating Text Files

Finding text sections

With the text editor, you can search for words or character strings in a text. Two functions are available:

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.
- ▶ Select the search function: Press the **FIND** soft key
- ▶ Press the **FIND CURRENT WORD** soft key
- ▶ Exit the search function: Press the **END** soft key

Finding any text

- ▶ Select the search function: Press the **FIND** soft key. The TNC displays the dialog prompt **Find text:**
- ▶ Enter the text that you wish to find
- ▶ Find the text: Press the **EXECUTE** soft key
- ▶ Exit the search function: Press the **END** soft key

11.9 Freely definable tables

Fundamentals

In freely definable tables you can read and save any information from the NC program. The Q parameter functions **FN 26** to **FN 28** are provided for this purpose.

You can change the format of freely definable tables, i.e. the columns and their properties, by using the structure editor. They enable you to make tables that are exactly tailored to your application.

You can also switch between table view (default setting) and form view.

M	Y	Z	A	C	DOC
1	100.001	49.999	0		PAT 1
1	99.994	49.999	0		PAT 2
2	99.990	50.001	0		PAT 3
3	100.002	49.995	0		PAT 4
4	99.990	50.003			PAT 5

Creating a freely definable table

- ▶ To call the file manager, press the **PGM MGT** key
- ▶ Enter any file name with the .TAB extension and 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** and confirm with the **ENT** key. The TNC opens a new table in the predefined format
- ▶ To adapt the table to your requirements you have to edit the table format, see "Editing the table format", page 428



Machine tool builders may define their own table templates and save them in the TNC. When you create a new table, the TNC opens a pop-up window listing all available table templates.



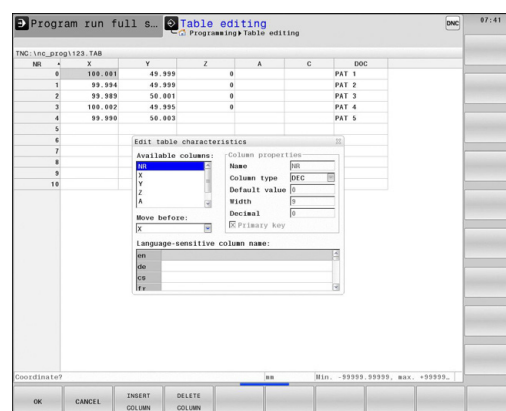
You can also save your own table templates in the TNC. To do this, you create a new table, change the table format and save the table in the **TNC:\system\proto** directory. Then your template will also be available in the list box for table templates when you create a new table.

11.9 Freely definable tables

Editing the table format

- Press the **EDIT FORMAT** soft key (shift the soft-key row): The TNC opens the editor form, in which the table structure is shown. The meanings of the structure commands (header entries) are shown in the following table.

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: Sign + or - BIN: Binary number DEC: Decimal, positive, complete number (cardinal number) HEX: Hexadecimal number INT: Complete 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
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



You can use a connected mouse or the TNC keyboard to navigate in the form. Navigation using the TNC keyboard:



- Press the navigation keys to go to the input fields. Use the arrow keys to navigate within an input field. To open pop-down menus, press the GOTO key.



In a table that already has lines, you cannot 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.

In a field of the **TSTAMP** column type you can reset an invalid value if you press the CE key and then the ENT key.

Exiting the structure editor

- Press the **OK** soft key. The TNC closes the editor form and applies the changes. All changes are discarded by pressing the **CANCEL** soft key.

Switching between table and form view

All tables with the file extension **.TAB** can be opened in either list view or form view.

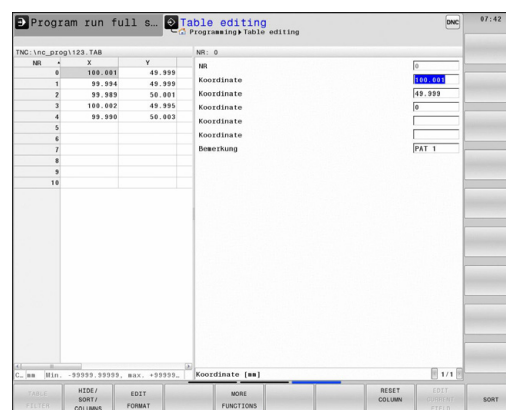


- Press the key for setting the screen layout. Select the respective soft key for list view or form view (form view: with or without dialog texts)

In the form view the TNC lists the line numbers with the contents of the first column in the left half of the screen.

In the right half you can change the data.

- Press the **ENT** key or the arrow key to move to the next input field.
- To select another line, press the green navigation key (folder symbol). This moves the cursor to the left window, and you can select the desired line with the arrow keys. Press the green navigation key to switch back to the input window.



11.9 Freely definable tables

FN 26: TABOPEN – Open a freely definable table

With the function **FN 26: TABOPEN** you open a freely definable table to be written to with **FN 27** or to be read from with **FN 28**.



Only one table can be open in an NC program. A new block with **FN 26: TABOPEN** automatically closes the last opened table.

The table to be opened must have the file name extension .TAB.

Example: Open the table TAB1.TAB, which is saved in the directory TNC:\DIR1.

```
56 FN 26: TABOPEN TNC:\DIR1\TAB1.TAB
```


FN 27: TABWRITE – Write to a freely definable table

With the **FN 27: TABWRITE** function you write to the table that you previously opened with **FN 26: TABOPEN**.

You can write several column names in a **TABWRITE** block. The column names must be written between quotation marks and separated by a comma. You define the values that the TNC is to write to the respective column with Q parameters.



Note that by default the **FN 27: TABWRITE** function writes values to the currently open table also in the Test run mode. The **FN18 ID992 NR16** function enables you to query in which operating mode the program is to be run. If the **FN27** function is to be run only in the **Program Run, Single Block** and **Program Run, Full Sequence** operating modes, you can skip the respective program section by using a jump command, page 307.

You can write only to numerical table fields.

If you wish to write to more than one column in a block, you must save the values under successive Q parameter numbers.

Example

You wish to write to the columns "Radius," "Depth" and "D" in line 5 of the presently opened table. The value to be written in the table must be saved in the Q parameters Q5, Q6 and Q7.

```
53 Q5 = 3.75
```

```
54 Q6 = -5
```

```
55 Q7 = 7.5
```

```
56 FN 27: TABWRITE 5/"RADIUS,DEPTH,D" = Q5
```


11.9 Freely definable tables

FN 28: TABREAD – Read from a freely definable table

With the **FN 28: TABREAD** function you read from the table previously opened with **FN 26: TABOPEN**.

You can define, i.e. read several column names in a **TABREAD** block. The column names must be written between quotation marks and separated by a comma. In the **FN 28** block you can define the Q parameter number in which the TNC is to write the value that is first read.



You can read only numerical table fields.
If you wish to read from more than one column in a block, the TNC will save the values under successive Q parameter numbers.

Example

You wish to read the values of the columns "Radius," "Depth" 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).

```
56 FN 28: TABREAD Q10 = 6/"RADIUS,DEPTH,D"
```


11.10 Dwell time FUNCTION FEED DWELL

Programming dwell time

Application



The behavior of this function varies depending on the respective machine.

Refer to your machine manual.

The **FUNCTION FEED DWELL** function is 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 not effective with rapid traverse and probing motion.



Damage to the workplace!

Do not use **FUNCTION FEED DWELL** for machining threads.

Procedure

Proceed as follows for the definition:

SPEC
FCT

- ▶ Show the soft-key row with special functions

PROGRAM
FUNCTIONS

- ▶ Select the menu for defining various plain-language functions

FUNCTION
FEED

- ▶ Select the **FUNCTION FEED** soft key

FEED
DWELL

- ▶ Select the **FEED DWELL** soft key
- ▶ Define the interval duration for dwelling D-TIME
- ▶ Define the interval duration for cutting F-TIME

NC block

13 FUNCTION FEED DWELL D-TIME0.5
F-TIME5

Programming: Special functions

11.10 Dwell time FUNCTION FEED DWELL

Resetting dwell time



Reset to the dwell time immediately following the machining with chip breaking.

NC block

18 FUNCTION FEED DWELL RESET

Use **FUNCTION FEED DWELL RESET** to reset the recurring dwell time.

Proceed as follows for the definition:

SPEC
FCT

- Show the soft-key row with special functions

PROGRAM
FUNCTIONS

- Select the menu for defining various plain-language functions

FUNCTION
FEED

- Select the **FUNCTION FEED** soft key

RESET
FEED
DWELL

- Select the **RESET FEED DWELL** soft key



You can also reset the dwell time by entering D-TIME 0.

The TNC automatically resets the **FUNCTION FEED DWELL** function at the end of a program.

12

**Programming:
Multiple Axis
Machining**

12.1 Functions for multiple axis machining

12.1 Functions for multiple axis machining

The TNC functions for multiple axis machining are described in this chapter.

TNC function	Description	Page
PLANE	Define machining in the tilted working plane	437
M116	Feed rate of rotary axes	461
PLANE/M128	Inclined-tool machining	459
FUNCTION TCPM	Define the behavior of the TNC when positioning the rotary axes (improvement of M128)	469
M126	Shortest-path traverse of rotary axes	462
M94	Reduce display value of rotary axes	463
M128	Define the behavior of the TNC when positioning the rotary axes	464
M138	Selection of tilted axes	467
M144	Calculate machine kinematics	468
LN blocks	Three-dimensional tool compensation	474

12.2 The PLANE Function: Tilting the working plane (Software Option 8)

Introduction



The machine manufacturer must enable the functions for tilting the working plane!

You can only use the **PLANE** function in its entirety on machines which have at least two rotary axes (head and/or table). Exception: **PLANE AXIAL** can also be used if only a single rotary axis is present or active on your machine.

The **PLANE** function is a powerful function for defining tilted working planes in various manners.

The parameter definition of the **PLANE** function is separated 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, see "Specifying the positioning behavior of the PLANE function", page 453



Danger of collision!

If you work with Cycle 8 **MIRROR IMAGE** in a tilted system, please note the following:

Program the tilting motion first and then define Cycle **8MIRROR IMAGE**.

Mirroring a rotary axis with Cycle **8** only mirrors the motions of the axis, but not the angles defined in the **PLANE** functions. As a result, the positioning of the axes changes.

Programs created on an iTNC 530 or on earlier TNCs are not compatible.



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 TNC automatically rescinds the radius compensation, which also rescinds the **M120** function.

Always use **PLANE RESET** to reset **PLANE** functions. Entering 0 in all **PLANE** parameters does not completely reset the function.


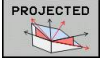
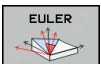
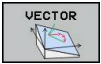
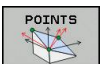

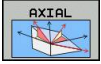

If you restrict the number of tilting axes with the **M138** function, your machine may provide only limited tilting possibilities.

The TNC only supports tilting the working plane with spindle axis Z.

12.2 The PLANE Function: Tilting the working plane (Software Option 8)

Overview

All **PLANE** functions available on the TNC describe the desired working plane independently of the rotary axes actually present on your machine. The following possibilities are available:

Soft key	Function	Required parameters	Page
	SPATIAL	Three spatial angles: SPA , SPB , and SPC	441
	PROJECTED	Two projection angles: PROPR and PROMIN and a rotation angle ROT	443
	EULER	Three Euler angles: precession (EULPR), nutation (EULNU) and rotation (EULROT),	444
	VECTOR	Normal vector for defining the plane and base vector for defining the direction of the tilted X axis	446
	POINTS	Coordinates of any three points in the plane to be tilted	448
	RELATIVE	Single, incrementally effective spatial angle	450
	AXIAL	Up to three absolute or incremental axis angles A,B,C	451
	RESET	Resetting the PLANE function	440

The PLANE Function: Tilting the working plane (Software Option 8) 12.2

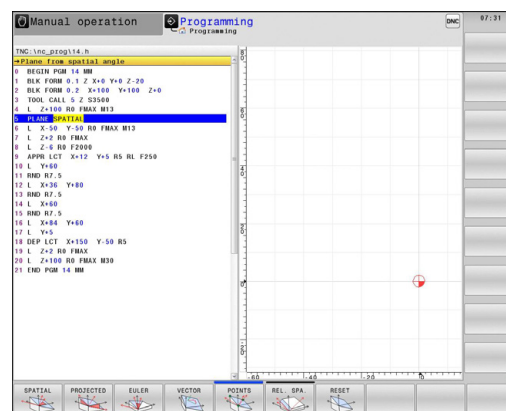
Defining the PLANE function

SPEC
FCT

- Show the soft-key row with special functions

TILT
MACHINING
PLANE

- Select the **PLANE** function: Press the **TILT MACHINING PLANE** soft key: The TNC displays the available definition possibilities in the soft-key row



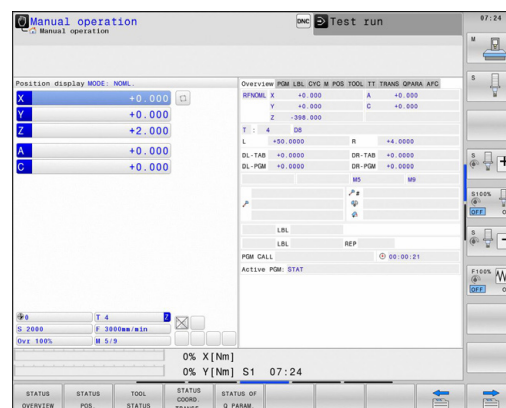
Selecting functions

- Select the desired function by soft key. The TNC continues the dialog and requests the required parameters

Position display

As soon as a **PLANE** function is active, the TNC shows the calculated spatial angle in the additional status display (see figure). As a rule, the TNC internally always calculates with spatial angles, independent of which **PLANE** function is active.

During tilting (**MOVE** or **TURN** mode) in the Distance-To-Go mode (**DIST**), the TNC shows (in the rotary axis) the distance to go (or calculated distance) to the final position of the rotary axis.



12.2 The PLANE Function: Tilting the working plane (Software Option 8)

Resetting the PLANE function

- ▶ Show the soft-key row with special functions
- ▶ To select special TNC functions, press the **SPECIAL TNC FUNCT.** soft key
- ▶ To select the PLANE function, press the **TILT MACHINING PLANE** soft key: The TNC displays the available definitions in the soft-key row
- ▶ Select the Reset function. This internally resets the **PLANE** function, but does not change the current axis positions
- ▶ Specify whether the TNC should automatically move the rotary axes to the default setting (**MOVE** or **TURN**) or not (**STAY**), see "Automatic positioning: MOVE/TURN/STAY (entry is mandatory)", page 453
- ▶ To conclude entry, Press END.

NC block

25 PLANE RESET MOVE ABST50 F1000



The **PLANE RESET** function resets the current **PLANE** function—or an active cycle **19**—completely (angles = 0 and function is inactive). It does not need to be defined more than once.

Deactivate tilting in the **Manual operation** operating mode in the 3D ROT menu.

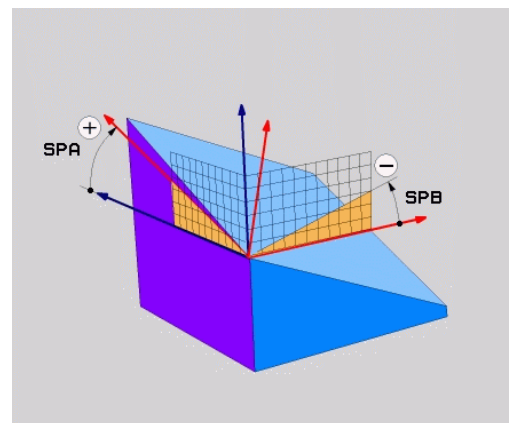
The PLANE Function: Tilting the working plane (Software Option 8) 12.2

Defining the working plane with the spatial angle: PLANE SPATIAL

Application

Spatial angles define a working plane using up to three rotations of the coordinate system; two perspectives that have always the same result are available for this purpose.

- **Rotations about the machine-based coordinate system:** The sequence of the rotations is first around the machine axis C, then around the machine axis B, and then around the machine axis A.
- **Rotations about the respectively tilted coordinate system:** The sequence of rotations is first around the machine axis C, then around the rotated axis B, and then around the rotated axis A. This perspective is usually easier to understand, because one rotary axis is fixed so that the rotations of the coordinate system are easier to comprehend.



Before programming, note the following

You must always define the three spatial angles **SPA**, **SPB**, and **SPC**, even if one of them = 0.

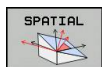
This operation corresponds to **19** if the entries in Cycle **19** are defined as spatial angles on the machine side.

PLANE SPATIAL is not permitted if Cycle 8 **MIRROR IMAGE** is active.

Parameter description for the positioning behavior: see "Specifying the positioning behavior of the PLANE function", page 453.

12.2 The PLANE Function: Tilting the working plane (Software Option 8)

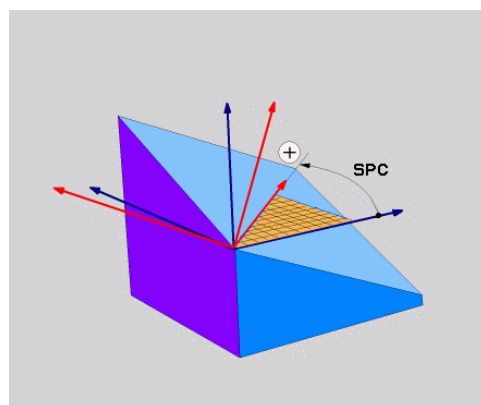
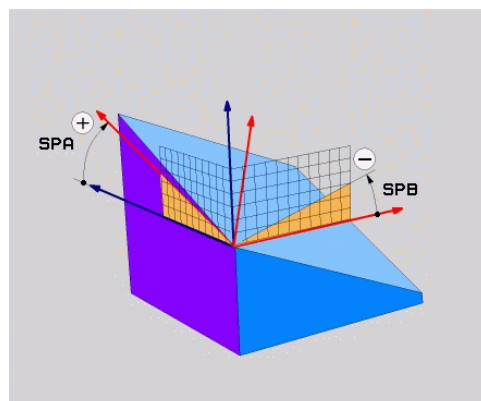
Input parameters



- ▶ **Spatial angle A?:** Rotational angle **SPA** around the fixed machine axis X (see figure at top right). Input range from -359.9999° to $+359.9999^\circ$
- ▶ **Spatial angle B?:** Rotational angle **SPB** around the fixed machine axis Y (see figure at top right). Input range from -359.9999° to $+359.9999^\circ$
- ▶ **Spatial angle C?:** Rotational angle **SPC** around the fixed machine axis Z (see figure at center right). Input range from -359.9999° to $+359.9999^\circ$
- ▶ Continue with the positioning properties, see "Specifying the positioning behavior of the PLANE function", page 453

Abbreviations used

Abbreviation	Meaning
SPATIAL	In space
SPA	S patial A : Rotation around the X axis
SPB	S patial B : Rotation around the Y axis
SPC	S patial C : Rotation around the Z axis



NC block

```
5 PLANE SPATIAL SPA+27 SPB+0 SPC
+45 .....
```


The PLANE Function: Tilting the working plane (Software Option 8) 12.2

Defining the working plane with the projection angle: PLANE PROJECTED

Application

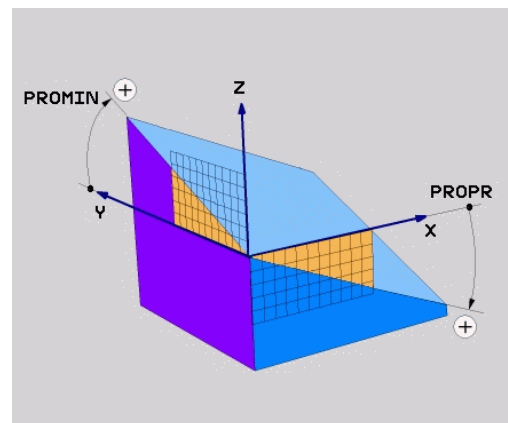
Projection angles define a machining plane through the entry of two angles that you determine by projecting the first coordinate plane (Z/X plane with tool axis Z) and the second coordinate plane (Y/Z with tool axis Z) onto the machining plane to be defined.



Before programming, note the following

You can only use projection angles if the angle definitions are given with respect to a rectangular cuboid. Otherwise there will be deformations on the workpiece.

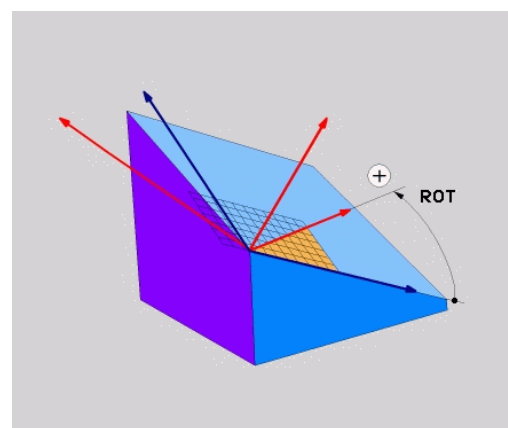
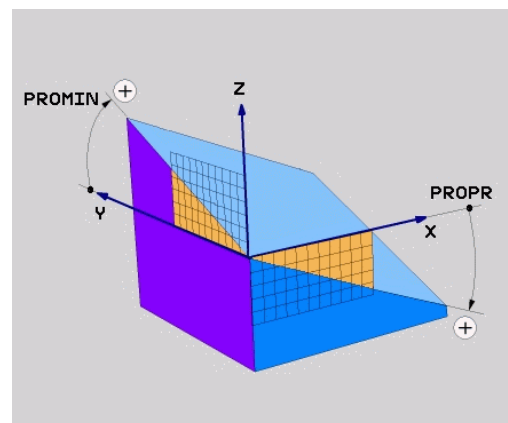
Parameter description for the positioning behavior: see "Specifying the positioning behavior of the PLANE function", page 453.



Input parameters



- ▶ **Proj. angle in 1st coord. plane?:** Projected angle of the tilted machining plane in the 1st coordinate plane of the fixed machine coordinate system (Z/X for tool axis Z, see figure at top right). Input range: from -89.9999° to $+89.9999^\circ$. The 0° axis is the principal axis of the active working plane (X for tool axis Z. See figure at top right for positive direction)
- ▶ **Proj. angle in 2nd coord. plane?:** Projected angle in the 2nd coordinate plane of the fixed machine coordinate system (Y/Z for tool axis Z, see figure at top right). Input range: from -89.9999° to $+89.9999^\circ$. 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; see figure at bottom right). Input range: -360° to $+360^\circ$
- ▶ Continue with the positioning properties, see "Specifying the positioning behavior of the PLANE function", page 453



NC block

```
5 PLANE PROJECTED PROPR+24 PROMIN+24 PROROT+30 .....
```


12.2 The PLANE Function: Tilting the working plane (Software Option 8)

Abbreviations used:

PROJECTED	Projected
PROPR	Principle plane
PROMIN	Minor plane
PROMIN	Rotation

Defining the working plane with the Euler angle: PLANE EULER

Application

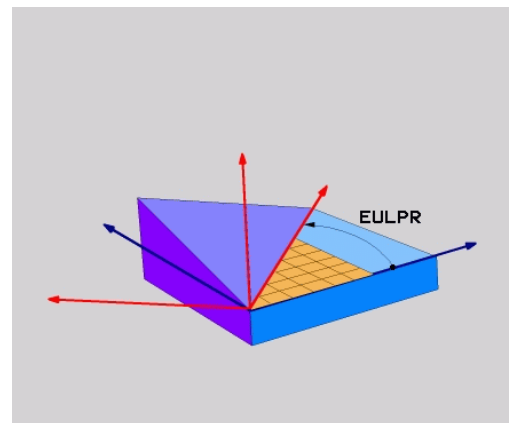
Euler angles define a machining plane through up to three **rotations about the respectively tilted coordinate system**. The Swiss mathematician Leonhard Euler defined these angles. When applied to the machine coordinate system, they have the following meanings:

Precession angle: EULPR	Rotation of the coordinate system around the Z axis
Nutation angle: EULNU	Rotation of the coordinate system around the X axis already shifted by the precession angle
Rotation angle: EULROT	Rotation of the tilted machining plane around the tilted Z axis



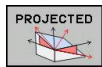
Before programming, note the following

Parameter description for the positioning behavior: see "Specifying the positioning behavior of the PLANE function", page 453.

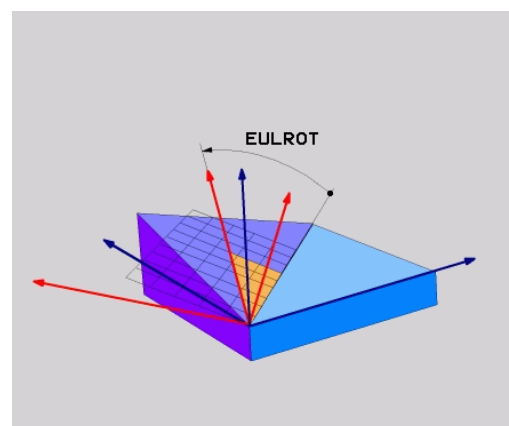
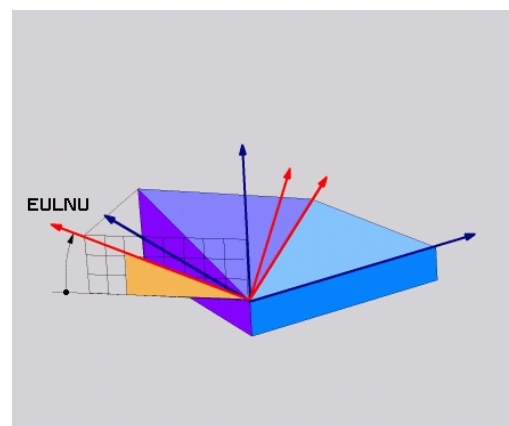
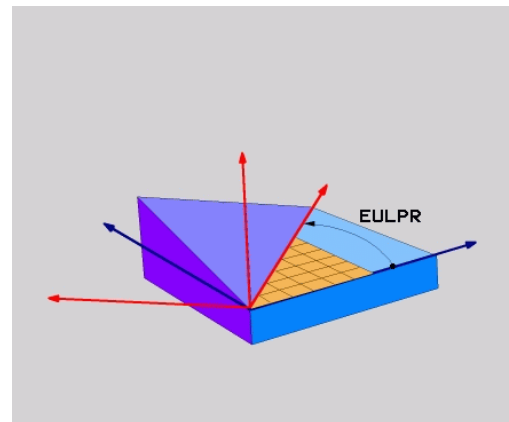


The PLANE Function: Tilting the working plane (Software Option 8) 12.2

Input parameters



- ▶ **Rot. angle of main coord. plane?:** Rotary angle **EULPR** around the Z axis (see figure at top right).
Please note:
 - Input range: -180.0000° to 180.0000°
 - The 0° axis is the X axis
- ▶ **Swivel angle of tool axis?:** Tilting angle **EULNU** of the coordinate system around the X axis shifted by the precession angle (see figure at center right).
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 in the tilted machining plane (see figure at bottom right).
Please note:
 - Input range: 0° to 360.0000°
 - The 0° axis is the X axis
- ▶ Continue with the positioning properties, see "Specifying the positioning behavior of the PLANE function", page 453



NC block

```
5 PLANE EULER EULPR45 EULNU20 EULROT22 .....
```


12.2 The PLANE Function: Tilting the working plane (Software Option 8)

Abbreviations used

Abbreviation	Meaning
EULER	Swiss mathematician who defined these angles
EULPR	P recession angle: angle describing the rotation of the coordinate system around the Z axis
EULNU	N utation angle: angle describing the rotation of the coordinate system around the X axis shifted by the precession angle
EULROT	R otation 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 TNC 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** (see figure at right). The normal vector is defined by the components **NX**, **NY** and **NZ**.

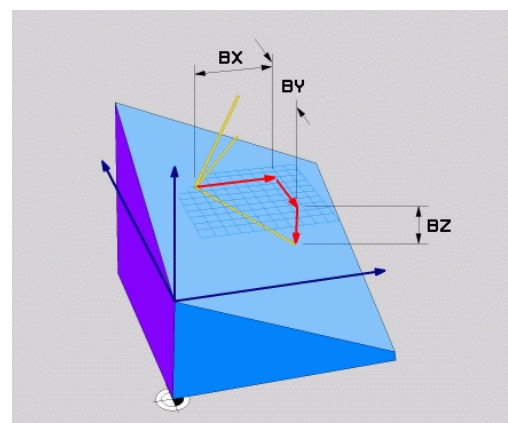


Before programming, note the following

The base vector defines the direction of the principal axis in the tilted machining plane, and the normal vector determines the orientation of the tilted machining plane, and at the same time is perpendicular to it.

The TNC calculates standardized vectors from the values you enter.

Parameter description for the positioning behavior: see "Specifying the positioning behavior of the PLANE function", page 453.

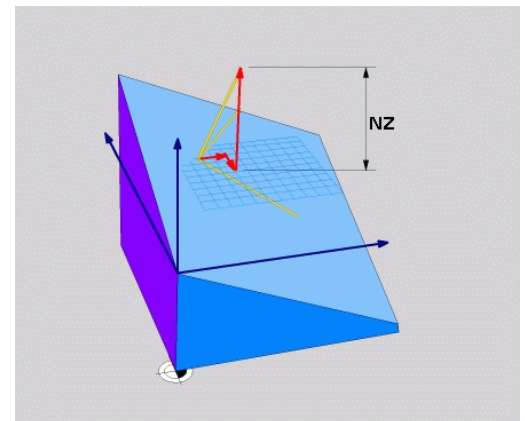
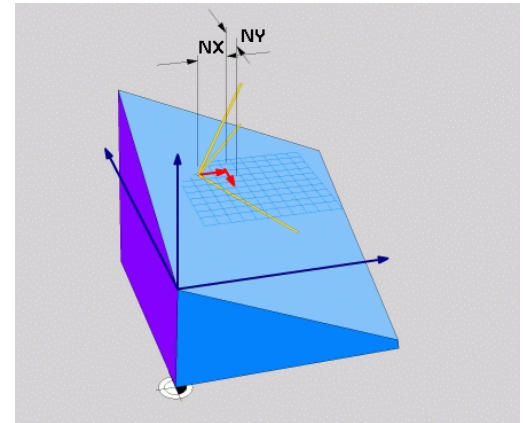
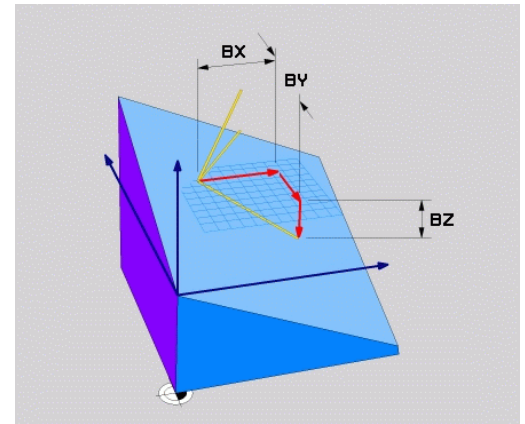


The PLANE Function: Tilting the working plane (Software Option 8) 12.2

Input parameters



- ▶ **X component of base vector?:** X component **BX** of the base vector B (see figure at top right). Input range: -9.9999999 to +9.9999999
- ▶ **Y component of base vector?:** Y component **BY** of the base vector B (see figure at top right). Input range: -9.9999999 to +9.9999999
- ▶ **Z component of base vector?:** Z component **BZ** of the base vector B (see figure at top right). Input range: -9.9999999 to +9.9999999
- ▶ **X component of normal vector?:** X component **NX** of the normal vector N (see figure at center right). Input range: -9.9999999 to +9.9999999
- ▶ **Y component of normal vector?:** Y component **NY** of the normal vector N (see figure at center right). Input range: -9.9999999 to +9.9999999
- ▶ **Z component of normal vector?:** Z component **NZ** of the normal vector N (see figure at lower right). Input range: -9.9999999 to +9.9999999
- ▶ Continue with the positioning properties, see "Specifying the positioning behavior of the PLANE function", page 453



NC block

```
5 PLANE VECTOR BX0.8 BY-0.4 BZ-0.42 NX0.2 NY0.2 NZ0.92 ..
```

Abbreviations used

Abbreviation	Meaning
VECTOR	Vector
BX, BY, BZ	Base vector: X , Y and Z components
NX, NY, NZ	Normal vector: X , Y and Z components

12.2 The PLANE Function: Tilting the working plane (Software Option 8)

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.



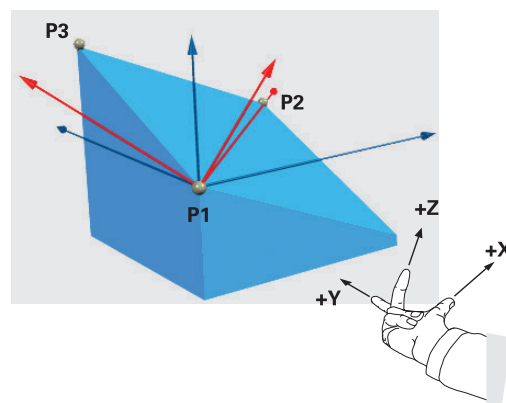
Before programming, note the following

The connection from Point 1 to Point 2 determines the direction of the tilted main axis (X for tool axis Z).

The direction of the tilted tool axis is determined by the position of Point 3 relative to the connecting line between Point 1 and Point 2. Use the right-hand rule (thumb = X axis, index finger = Y axis, middle finger = Z axis (see figure at right)) to remember: thumb (X axis) points from Point 1 to Point 2, index finger (Y axis) points parallel to the tilted Y axis in the direction of Point 3. Then the middle finger points in the direction of the tilted tool axis.

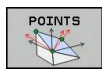
The three points define the slope of the plane. The position of the active datum is not changed by the TNC.

Parameter description for the positioning behavior: see "Specifying the positioning behavior of the PLANE function", page 453.

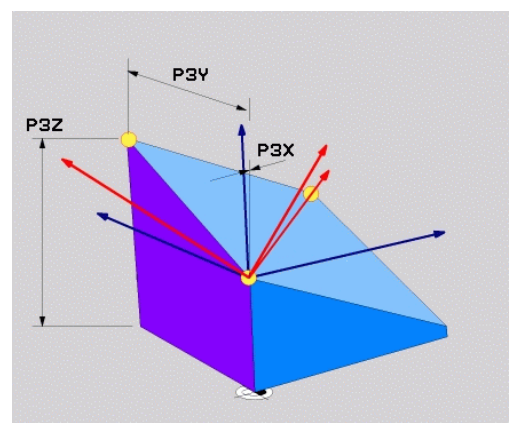
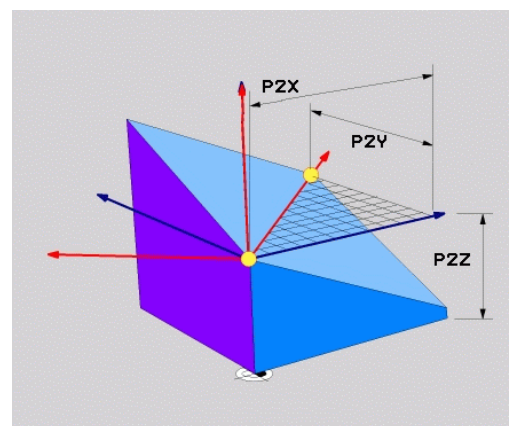
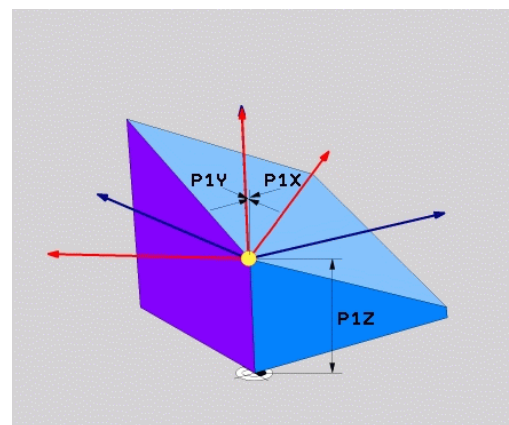


The PLANE Function: Tilting the working plane (Software Option 8) 12.2

Input parameters



- ▶ **X coordinate of 1st plane point?:** X coordinate **P1X** of the 1st plane point (see figure at top right)
- ▶ **Y coordinate of 1st plane point?:** Y coordinate **P1Y** of the 1st plane point (see figure at top right)
- ▶ **Z coordinate of 1st plane point?:** Z coordinate **P1Z** of the 1st plane point (see figure at top right)
- ▶ **X coordinate of 2nd plane point?:** X coordinate **P2X** of the 2nd plane point (see figure at center right)
- ▶ **Y coordinate of 2nd plane point?:** Y coordinate **P2Y** of the 2nd plane point (see figure at center right)
- ▶ **Z coordinate of 2nd plane point?:** Z coordinate **P2Z** of the 2nd plane point (see figure at center right)
- ▶ **X coordinate of 3rd plane point?:** X coordinate **P3X** of the 3rd plane point (see figure at bottom right)
- ▶ **Y coordinate of 3rd plane point?:** Y coordinate **P3Y** of the 3rd plane point (see figure at bottom right)
- ▶ **Z coordinate of 3rd plane point?:** Z coordinate **P3Z** of the 3rd plane point (see figure at bottom right)
- ▶ Continue with the positioning properties, see "Specifying the positioning behavior of the PLANE function", page 453



NC block

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

Abbreviations used

Abbreviation Meaning

Abbreviation	Meaning
POINTS	Points

12.2 The PLANE Function: Tilting the working plane (Software Option 8)

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

Application

Use an incremental 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.



Before programming, note the following

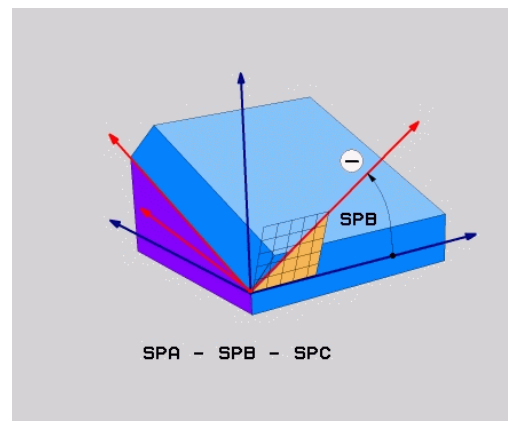
The defined angle is always in effect in respect to the active working plane, regardless of the function you have used to activate it.

You can program any number of **PLANE RELATIVE** functions in a row.

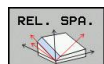
If you want to return to the working plane that was active before the **PLANE RELATIVE** function, define the **PLANE RELATIVE** function again with the same angle but with the opposite algebraic sign.

If you use the **PLANE RELATIVE** function in a non-tilted working plane, then you simply rotate the non-tilted plane about the spatial angle defined in the **PLANE** function.

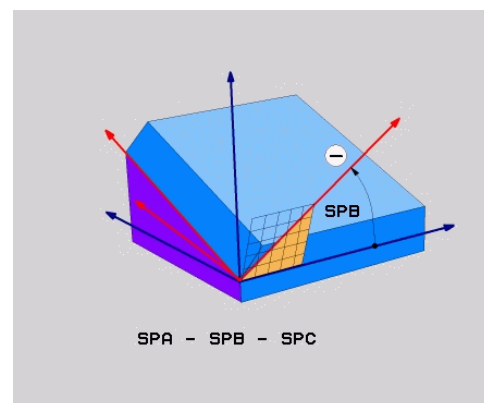
Parameter description for the positioning behavior: see "Specifying the positioning behavior of the PLANE function", page 453.



Input parameters



- **Incremental angle?:** Spatial angle about which the active machining plane is to be rotated additionally (see figure at right). Use a soft key to select the axis to be rotated about. Input range: -359.9999° to +359.9999°
- Continue with the positioning properties, see "Specifying the positioning behavior of the PLANE function", page 453



Abbreviations used

Abbreviation	Meaning
RELATIVE	Relative to

NC block

```
5 PLANE RELATIV SPB-45 .....
```


The PLANE Function: Tilting the working plane (Software Option 8) 12.2

Tilting the working plane through axis angle: PLANE AXIAL (FCL 3 function)

Application

The **PLANE AXIAL** function defines both the position of the working plane and the nominal coordinates of the rotary axes. This function is particularly easy to use on machines with Cartesian coordinates and with kinematics structures in which only one rotary axis is active.



PLANE AXIAL can also be used if you have only one rotary axis active on your machine.

You can use the **PLANE RELATIVE** function after **PLANE AXIAL** if your machine allows spatial angle definitions. Refer to your machine manual.



Before programming, note the following

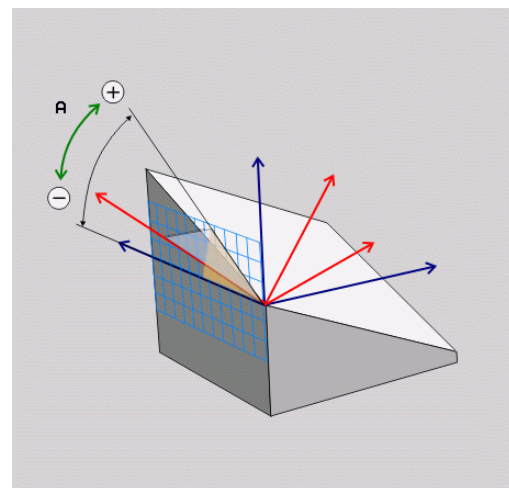
Enter only axis angles that actually exist on your machine. Otherwise the TNC generates an error message.

Rotary axis coordinates defined with **PLANE AXIAL** are modally effective. Successive definitions therefore build on each other. Incremental input is allowed.

Use **PLANE RESET** to reset the **PLANE AXIAL** function. Resetting by entering 0 does not deactivate **PLANE AXIAL**.

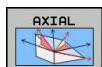
SEQ, **TABLE ROT** and **COORD ROT** have no function in conjunction with **PLANE AXIAL**.

Parameter description for the positioning behavior: see "Specifying the positioning behavior of the PLANE function", page 453.

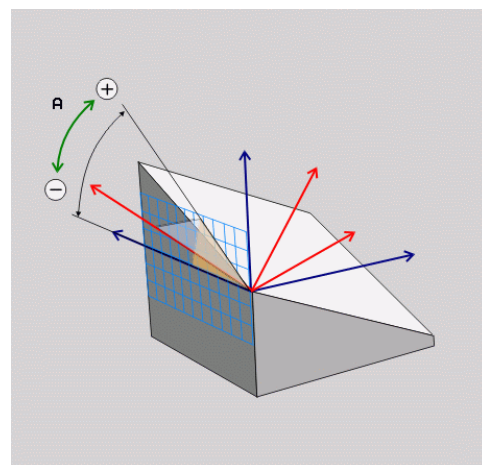


12.2 The PLANE Function: Tilting the working plane (Software Option 8)

Input parameters



- ▶ **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 $+99999.9999^{\circ}$
- ▶ **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 $+99999.9999^{\circ}$
- ▶ **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 $+99999.9999^{\circ}$
- ▶ Continue with the positioning properties, see "Specifying the positioning behavior of the PLANE function", page 453



NC block

5 PLANE AXIAL B-45

Abbreviations used

Abbreviation	Meaning
AXIAL	In the axial direction

The PLANE Function: Tilting the working plane (Software Option 8) 12.2

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
- Selection of alternate tilting possibilities (not with **PLANE AXIAL**)
- Selection of the type of transformation (not with **PLANE AXIAL**)



Danger of collision!

If you work with Cycle 8 **MIRROR IMAGE** in a tilted system, please note the following:

Program the tilting motion first and then define Cycle 8 **8MIRROR IMAGE**.

Mirroring a rotary axis with Cycle **8** only mirrors the motions of the axis, but not the angles defined in the PLANE functions. As a result, the positioning of the axes changes.

Programs created on an iTNC 530 or on earlier TNCs are not compatible.

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

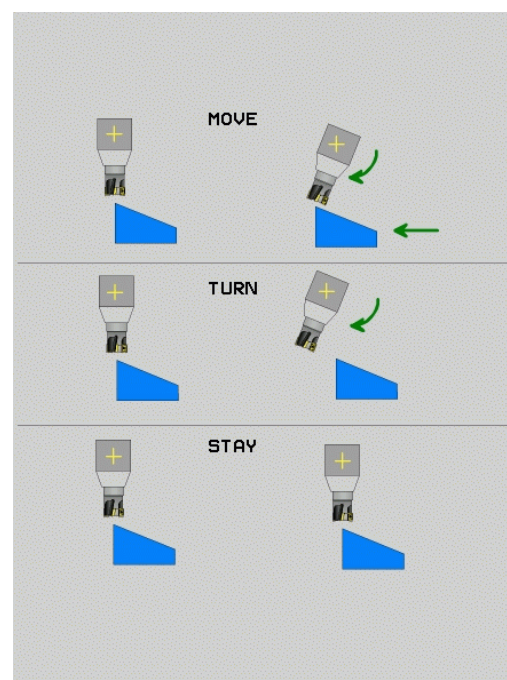
After you have entered all parameters for the plane definition, you must specify how the rotary axes will be positioned to the calculated axis values:

- | | |
|---|--|
| <div style="border: 1px solid black; padding: 2px; width: fit-content; margin-bottom: 10px;">MOVE</div> | <p>► 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 TNC carries out a compensation movement in the linear axes</p> |
| <div style="border: 1px solid black; padding: 2px; width: fit-content; margin-bottom: 10px;">TURN</div> | <p>► The PLANE function is to automatically position the rotary axes to the calculated position values, but only the rotary axes are positioned. The TNC does not carry out a compensation movement in the linear axes</p> |
| <div style="border: 1px solid black; padding: 2px; width: fit-content; margin-bottom: 10px;">STAY</div> | <p>► You will position the rotary axes later in a separate positioning block</p> |

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 numerical value, you can also position with **FMAX** (rapid traverse) or **FAUTO** (feed rate from the **TOOL CALLT** block).



12 Programming: Multiple Axis Machining

12.2 The PLANE Function: Tilting the working plane (Software Option 8)



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

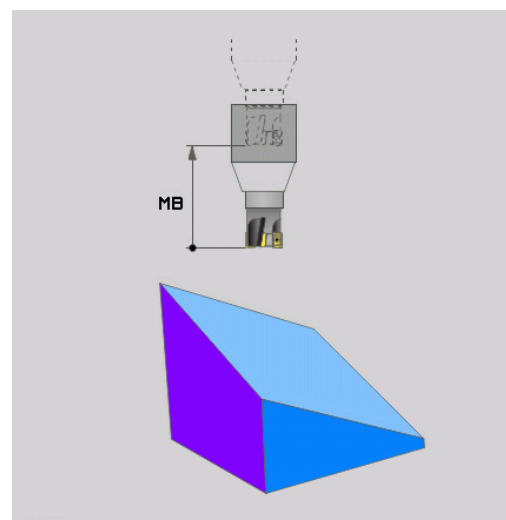
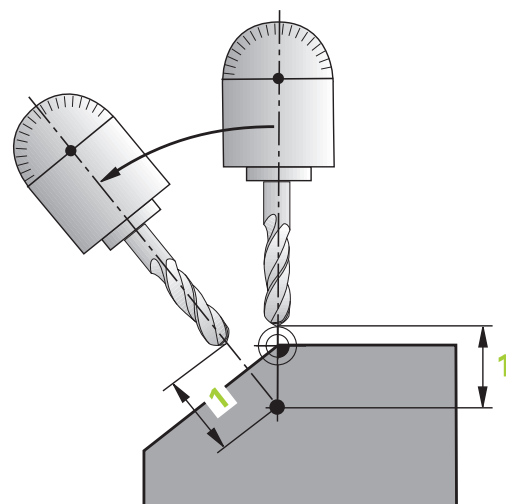
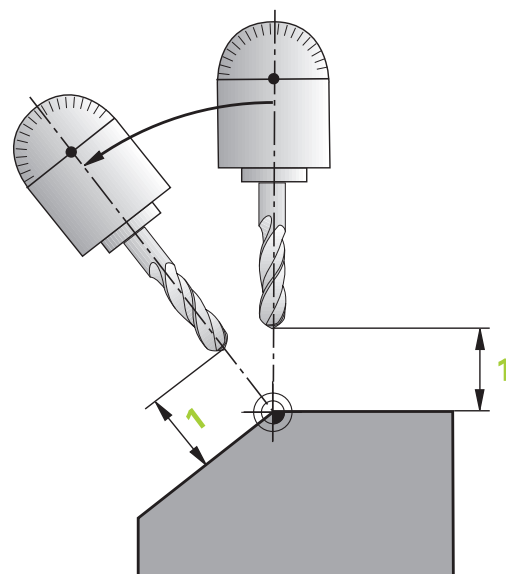
- ▶ **Dist. tool tip - center of rot.** (incremental): The TNC tilts the tool (or table) relative to the tool tip. The **DIST** parameter shifts the center of rotation of the positioning movement relative to the current position of the tool tip.



Note:

- If the tool is already at the given distance to the workpiece before positioning, then relatively speaking the tool is at the same position after positioning (see figure at center right, **1** = DIST).
- If the tool is not at the given distance to the workpiece before positioning, then relatively speaking the tool is offset from the original position after positioning (see figure at bottom right, **1**=DIST).

- ▶ **Feed rate? F=:** Contour speed at which the tool should be positioned
- ▶ **Retraction length in the tool axis?:** Retraction path **MB** is effective incrementally from the current tool position in the active tool axis direction that the TNC approaches **before tilting**. **MB MAX** positions the tool just before the software limit switch.



The PLANE Function: Tilting the working plane (Software Option 8) 12.2

Positioning the rotary axes in a separate block

Proceed as follows if you want to position the rotary axes in a separate positioning block (option **STAY** selected):



Danger of collision!

Pre-position the tool to a position where there is no danger of collision with the workpiece (clamping devices) during positioning.

Do not program mirroring of the rotary axis between the PLANE function and the positioning, otherwise the control positions to the mirrored values but the PLANE function calculates without mirroring.

- ▶ Select any **PLANE** function, and define automatic positioning with the **STAY** option. During program execution the TNC 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 TNC

NC example blocks: Position a machine with a rotary table C and a tilting table A to a space angle of B+45°

...	
12 L Z+250 R0 FMAX	Position at clearance height
13 PLANE SPATIAL SPA+0 SPB+45 SPC+0 STAY	Define and activate the PLANE function
14 L A+Q120 C+Q122 F2000	Position the rotary axis with the values calculated by the TNC
...	Define machining in the tilted working plane

12

Programming: Multiple Axis Machining

12.2 The PLANE Function: Tilting the working plane (Software Option 8)

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

The position you define for the working plane is used by the TNC to calculate the appropriate positioning of the rotary axes present on the machine. In general there are always two solution possibilities.

Use the **SEQ** switch to specify which possibility the TNC should use:

- **SEQ+** positions the master axis so that it assumes a positive angle. The master axis is the 1st rotary axis from the tool, or the last rotary axis from the table (depending on the machine configuration (see figure at top right)).
- **SEQ-** positions the master axis so that it assumes a negative angle.

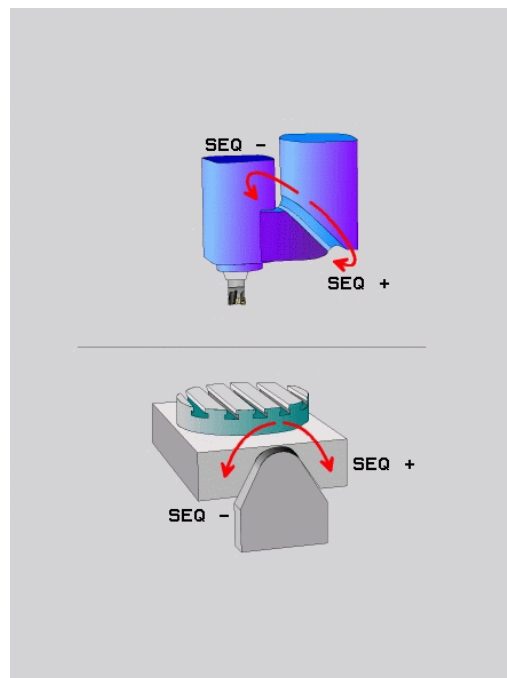
If the solution you chose with **SEQ** is not within the machine's range of traverse, the TNC displays the **Entered angle not permitted** error message.



When the **PLANE AXIS** function is used, the **SEQ** switch is nonfunctional.

If you do not define **SEQ**, the TNC determines the solution as follows:

- 1 The TNC first checks whether both solution possibilities are within the traverse range of the rotary axes.
- 2 If they are, then the TNC selects the shortest possible solution.
- 3 If only one solution is within the traverse range, the TNC selects this solution
- 4 If neither solution is within the traverse range, the TNC displays the **Entered angle not permitted** error message.



The PLANE Function: Tilting the working plane (Software Option 8) 12.2

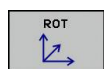
Example for a machine with a rotary table C and a tilting table

A. Programmed function: PLANE SPATIAL SPA+0 SPB+45 SPC+0

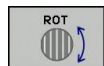
Limit switch	Starting position	SEQ	Resulting axis position
None	A+0, C+0	not prog.	A+45, C+90
None	A+0, C+0	+	A+45, C+90
None	A+0, C+0	–	A–45, C–90
None	A+0, C–105	not prog.	A–45, C–90
None	A+0, C–105	+	A+45, C+90
None	A+0, C–105	–	A–45, C–90
$-90 < A < +10$	A+0, C+0	not prog.	A–45, C–90
$-90 < A < +10$	A+0, C+0	+	Error message
None	A+0, C–135	+	A+45, C+90

Selecting the type of transformation (entry optional)

For tilting angles that only rotate the coordinate system around the tool axis, a specific function enables you to define the type of transformation:



- **COORD ROT** specifies that the PLANE function should only rotate the coordinate system to the defined tilting angle. Compensation results by computing and a rotary axis is not moved



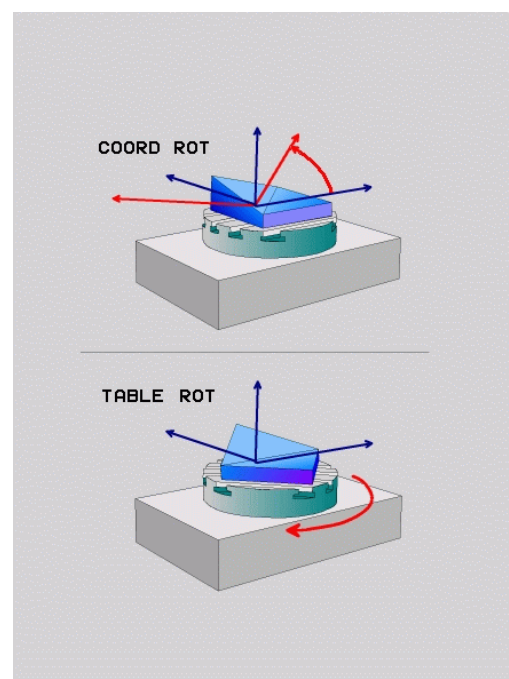
- **TABLE ROT** specifies that the PLANE function should position the rotary axes to the defined tilting angle. Compensation results from rotating the workpiece.



When the **PLANE AXIAL** function is used, **COORD ROT** and **TABLE ROT** are nonfunctional.

COORD ROT is active only if tilting is around the tool axis only, e. g. **SPC+45** with tool axis **Z**. As soon as a second swivel axis is required for implementation, **TABLE ROT** is automatically active.

If you use the **TABLE ROT** function in conjunction with a basic rotation and a tilting angle of 0, then the TNC tilts the table to the angle defined in the basic rotation.



12 Programming: Multiple Axis Machining

12.2 The PLANE Function: Tilting the working plane (Software Option 8)

Tilt the working plane without rotary axes



This feature must be enabled and adapted by the machine tool builder. Refer to your machine manual.

The machine tool builder must take into account e.g. the precise angle of a mounted angular head in the kinematics description.

You can also align the programmed working plane perpendicular to the tool without rotary axes, e.g. for 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:

NC syntax

```
TOOL CALL 5 Z S4500
```

```
PLANE SPATIAL SPA+0 SPB-90 SPC+0 STAY
```



The swivel angle must be precisely adapted to the tool angle, otherwise the TNC outputs an error message.

12.3 Inclined-tool machining in a tilted plane (Option #9)

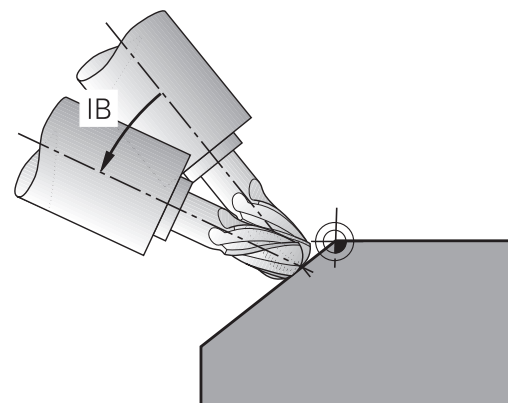
Function

In combination with **M128** and the new **PLANE** functions, **inclined-tool machining** in a tilted machining plane is now possible. Two possibilities are available for definition:

- Inclined-tool machining via incremental traverse of a rotary axis
- Inclined-tool machining via normal vectors



Inclined-tool machining in a tilted machining plane only functions with spherical cutters. With 45° swivel heads and tilting tables you can also define the incline angle as a space angle. Use the **TCPM FUNCTION**, see "FUNCTION TCPM (Option #9)", page 469.



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 NC blocks

...	
12 L Z+50 R0 FMAX	Position at clearance height
13 PLANE SPATIAL SPA+0 SPB-45 SPC+0 MOVE ABST50 F1000	Define and activate the PLANE function
14 M128	Activate M128
15 L IB-17 F1000	Set the incline angle
...	Define machining in the tilted working plane

12 Programming: Multiple Axis Machining

12.3 Inclined-tool machining in a tilted plane (Option #9)

Inclined-tool machining via normal vectors



Only one directional vector can be defined in the **LN** block. This vector defines the incline angle (normal vector **NX**, **NY**, **NZ**, or tool direction vector **TX**, **TY**, **TZ**).

- ▶ Retract the tool
- ▶ Define any PLANE function; consider the positioning behavior
- ▶ Activate M128
- ▶ Execute program with LN blocks in which the tool direction is defined by a vector

Example NC blocks

...	
12 L Z+50 R0 FMAX	Position at clearance height
13 PLANE SPATIAL SPA+0 SPB+45 SPC+0 MOVE ABST50 F1000	Define and activate the PLANE function
14 M128	Activate M128
15 LN X+31.737 Y+21.954 Z+33.165 NX+0.3 NY+0 NZ+0.9539 F1000 M3	Set the incline angle with the normal vector
...	Define machining in the tilted working plane

12.4 Miscellaneous functions for rotary axes

Feed rate in mm/min on rotary axes A, B, C: M116 (Option #8)

Standard behavior

The TNC 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 axis rotation.

The larger this distance becomes, the greater the contouring feed rate.

Feed rate in mm/min on rotary axes with M116



The machine geometry must be specified by the machine tool builder in the description of kinematics. M116 works only on rotary tables. M116 cannot be used with swivel heads. If your machine is equipped with a table/head combination, the TNC ignores the swivel-head rotary axes.

M116 is also effective in an active tilted working plane and in combination with M128 if you used the **M138** function to select rotary axes, see "Selecting tilting axes: M138", page 467. Then **M116** affects only those rotary axes that were selected with **M138**.

The TNC interprets the programmed feed rate of a rotary axis in degrees/min (or 1/10 inch/min). In this case, the TNC calculates the feed for the block at the start of each block. With a rotary axis, the feed rate is not changed during execution of the block even if the tool moves toward the center of the rotary axis.

Effect

M116 is effective in the working plane. To reset M116, enter M117. M116 is also canceled at the end of the program.

M116 becomes effective at the start of block.

12.4 Miscellaneous functions for rotary axes

Shortest-path traverse of rotary axes: M126

Standard behavior



The behavior of the TNC when positioning the rotary axes depends on the machine tool. Refer to your machine manual.

The standard behavior of the TNC while positioning rotary axes whose display has been reduced to values less than 360° is dependent on machine parameter **shortestDistance** (300401). This machine parameter defines whether the TNC should consider the difference between nominal and actual position, or whether it should always (even without M126) choose the shortest path to the programmed position. Examples:

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

Behavior with M126

With M126, the TNC will move the axis on the shorter path of traverse for rotary axes whose display is reduced to values less than 360°. Examples:

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

Effect

M126 becomes effective at the start of 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 TNC 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 TNC first reduces the current angular value to a value less than 360° and then moves the tool to the programmed value. If several rotary axes are active, M94 will reduce the display of all rotary axes. As an alternative you can enter a rotary axis after M94. The TNC then reduces the display only of this axis.

Example NC blocks

To reduce display of all active rotary axes:

```
L M94
```

To reduce display of the C axis only:

```
L M94 C
```

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

```
L C+180 FMAX M94
```

Effect

M94 is effective only in the block in which it is programmed.

M94 becomes effective at the start of block.

12 Programming: Multiple Axis Machining

12.4 Miscellaneous functions for rotary axes

Maintaining the position of the tool tip when positioning with tilted axes (TCPM): M128 (Option #9)

Standard behavior

The TNC moves the tool to the positions given in the machining program. If the position of a tilted axis changes in the program, the resulting offset in the linear axes must be calculated, and traversed in a positioning block.

Behavior with M128 (TCPM: Tool Center Point Management)



The machine geometry must be specified by the machine tool builder in the description of kinematics.

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



Caution: Danger to the workpiece!

For tilted axes with Hirth coupling: Do not change the position of the tilted axis until after retracting the tool. Otherwise you might damage the contour when disengaging from the coupling.

After **M128** you can program another feed rate, at which the TNC will carry out the compensation movements in the linear axes.

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

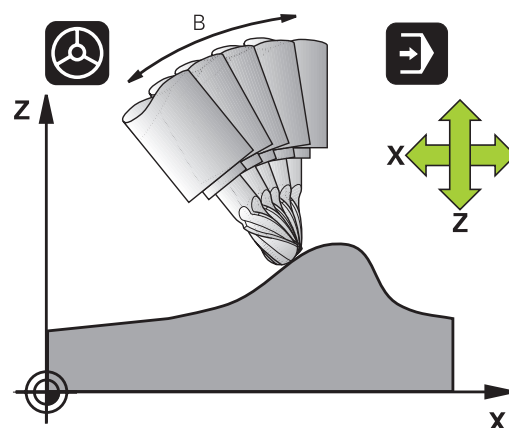


Before positioning with **M91** or **M92** and before a **TOOL CALL**, **RESET M128**.

To avoid contour gouging you must use only spherical cutters with **M128**.

The tool length must refer to the spherical center of the tool tip.

If **M128** is active, the TNC shows the TCPM symbol in the status display.



M128 on tilting tables

If you program a tilting table movement while **M128** is active, the TNC rotates the coordinate system accordingly. If, for example, you rotate the C axis by 90° (through a positioning command or datum shift) and then program a movement in the X axis, the TNC executes the movement in the machine axis Y.

The TNC also transforms the defined datum, 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 **RL/RR**, the TNC will automatically position the rotary axes for certain machine geometrical configurations (Peripheral milling, see "Three-dimensional tool compensation (Option #9)", page 474).

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 TNC also cancels **M128** if you select a new program in a program run operating mode.

Example NC blocks

Feed rate of 1000 mm/min for compensation movements:

```
L X+0 Y+38.5 IB-15 RL F125 M128 F1000
```


12.4 Miscellaneous functions for rotary axes

Inclined machining with noncontrolled rotary axes

If you have noncontrolled rotary axes (counting axes) on your machine, then in combination with M128 you can also perform inclined machining operations with these axes.

- 1 Manually traverse the rotary axes to the desired positions.
M128 must not be active!
- 2 Activate M128: The TNC reads the actual values of all rotary axes present, calculates from this the new position of the tool center point, and updates the position display
- 3 The TNC performs the necessary compensating movement in the next positioning block
- 4 Carry out the machining operation
- 5 At the end of program, reset M128 with M129, and return the rotary axes to the initial positions

Proceed as follows:



As long as M128 is active, the TNC monitors the actual positions of the noncontrolled rotary axes. If the actual position deviates from the nominal position by a value greater than that defined by the machine manufacturer, the TNC outputs an error message and interrupts program run.

Selecting tilting axes: M138

Standard behavior

The TNC performs M128, TCPM, and tilts the working plane, only in those axes for which the machine tool builder has set the appropriate machine parameters.

Behavior with M138

The TNC performs the above functions only in those tilting axes that you have defined using M138.



If you restrict the number of tilting axes with the **M138** function, your machine may provide only limited tilting possibilities.

Effect

M138 becomes effective at the start of block.

You can reset M138 by reprogramming it without entering any axes.

Example NC blocks

Perform the above-mentioned functions only in the tilting axis C:

```
L Z+100 R0 FMAX M138 C
```


12.4 Miscellaneous functions for rotary axes

Compensating the machine's kinematics configuration for ACTUAL/NOMINAL positions at end of block: M144 (Option #9)

Standard behavior

The TNC moves the tool to the positions given in the machining program. If the position of a tilted axis changes in the program, the resulting offset in the linear axes must be calculated, and traversed in a positioning block.

Behavior with M144

The TNC calculates into the position value any changes in the machine's kinematics configuration which result, for example, from adding a spindle attachment. If the position of a controlled tilted axis changes, the position of the tool tip to the workpiece is also changed. The resulting offset is calculated in the position display.



Positioning blocks with M91/M92 are permitted if M144 is active.

The position display in the operating modes FULL SEQUENCE and SINGLE BLOCK 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 function in connection with M128 or a tilted working plane.

You can cancel M144 by programming M145.



The machine geometry must be specified by the machine tool builder in the description of kinematics.

The machine tool builder determines the behavior in the automatic and manual operating modes. Refer to your machine manual.

12.5 FUNCTION TCPM (Option #9)

Function



The machine geometry must be specified by the machine tool builder in the description of kinematics.



For tilted axes with Hirth coupling:

Only change the position of the tilted axis after retracting the tool. Otherwise you might damage the contour when disengaging from the coupling.

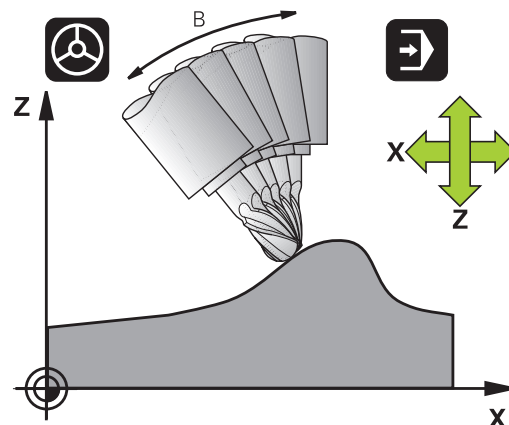


Before positioning with **M91** or **M92** and before a **TOOL CALL**: Reset **TCPM FUNCTION**.

To avoid contour gouging you must use only spherical cutters with **TCPM FUNCTION**.

The tool length must refer to the spherical center of the tool tip.

If **TCPM FUNCTION** is active, the TNC shows the symbol **TCPM** in the position display.



TCPM FUNCTION is an improvement on the **M128** function, with which you can define the behavior of the TNC when positioning the rotary axes. In contrast to **M128**, with **TCPM FUNCTION** you can define the mode of action of various functions:

- Mode of action of the programmed feed rate: **F TCP / F CONT**
- Interpretation of the rotary axis coordinates programmed in the NC program: **AXIS POS / AXIS SPAT**
- Type of interpolation between start and target position: **PATHCTRL AXIS / PATHCTRL VECTOR**

Defining the TCPM FUNCTION

SPEC
FCT

- ▶ Select the special functions

PROGRAM
FUNCTIONS

- ▶ Select the programming aids

FUNCTION
TCPM

- ▶ Select the TCPM FUNCTION

12.5 FUNCTION TCPM (Option #9)

Mode of action of the programmed feed rate

The TNC provides two functions for defining the mode of action of the programmed feed rate:

- | | |
|---|---|
| <div style="border: 1px solid black; padding: 2px; width: fit-content;">F
TCP</div> | <p>► F TCP determines that the programmed feed rate is interpreted as the actual relative velocity between the tool point (TCP = Tool Center Point) and the workpiece.</p> |
| <div style="border: 1px solid black; padding: 2px; width: fit-content;">F
CONTOUR</div> | <p>► F CONT determines that the programmed feed rate is interpreted as the contouring feed rate of the axis programmed in the respective NC block.</p> |

Example NC blocks

...	
13 FUNCTION TCPM F TCP ...	Feed rate refers to the tool tip
14 FUNCTION TCPM F CONT ...	Feed rate is interpreted as the speed of the tool along the contour
...	

Interpretation of the programmed rotary axis coordinates

Up to now, machines with 45° swivel heads or 45° tilting tables could not easily set the angle of inclination or a tool orientation with respect to the currently active coordinate system (spatial angle). This function could only be realized through specially written programs with normal vectors (LN blocks).

The TNC now provides the following function:

- | | |
|---|--|
| <div style="border: 1px solid black; padding: 2px; width: fit-content;">AXIS
POSITION</div> | <p>► AXIS POS determines that the TNC interprets the programmed coordinates of rotary axes as the nominal position of the respective axis</p> |
| <div style="border: 1px solid black; padding: 2px; width: fit-content;">AXIS
SPATIAL</div> | <p>► AXIS SPAT determines that the TNC interprets the programmed coordinates of rotary axes as the spatial angle</p> |



AXIS POS should be used primarily if your machine is equipped with Cartesian rotary axes. You can also use **AXIS POS** with 45°-swivel heads/tilting tables if it is ensured that the programmed rotary axis coordinates define the desired orientation of the working plane correctly (this can be accomplished with a CAM system, for example).

AXIS SPAT: The rotary axis coordinates entered in the positioning block are space angles that are given with respect to the currently active (perhaps tilted) coordinate system (incremental space angle).

After you switch on **FUNCTION TCPM** with **AXIS SPAT**, in the first positioning block you should always program all three spatial angles in the inclination angle definition. This also applies if one or more spatial angles are 0°. **AXIS SPAT:** The rotary axis coordinates entered in the positioning block are space angles that are given with respect to the currently active (perhaps tilted) coordinate system (incremental space angle).

Example NC blocks

...	
13 FUNCTION TCPM F TCP AXIS POS ...	Rotary axis coordinates are axis angles
...	
18 FUNCTION TCPM F TCP AXIS SPAT ...	Rotary axis coordinates are spatial angles
20 L A+0 B+45 C+0 F MAX	Set tool orientation to B+45 degrees (spatial angle). Define space angle A and C with 0
...	

12.5 FUNCTION TCPM (Option #9)

Type of interpolation between the starting and end position

The TNC provides two functions for defining the type of interpolation between the starting and end position:

- PATH
CONTROL
AXIS

► **PATHCTRL AXIS** determines that the tool point between the starting and end position of the respective NC block moves on a straight line (**Face Milling**). The direction of the tool axis at the starting and end positions corresponds to the respective programmed values, but the tool circumference does not describe a defined path between the starting and end positions. The surface produced by milling with the tool circumference (**Peripheral Milling**) depends on the machine geometry
- PATH
CONTROL
VECTOR

► **PATHCTRL VECTOR** determines that the tool tip between the starting and end position of the respective NC block moves on a straight line and also that the direction of the tool axis between starting and end position is interpolated so that a plane results from machining at the tool circumference (**Peripheral Milling**)



With PATHCTRL VECTOR, remember:

Any defined tool orientation is generally accessible through two different tilting angle positions. The TNC uses the solution over the shortest available path—starting from the current position.

To attain the most continuous multiaxis movement possible, define Cycle 32 with a **tolerance for rotary axes** (see Touch Probe Cycles User's Manual, Cycle 32 TOLERANCE). The tolerance of the rotary axes should be about the same as the tolerance of the contouring deviation that is also defined in Cycle 32. The greater the tolerance for the rotary axes is defined, the greater are the contour deviations during peripheral milling.


Example NC blocks

...	
13 FUNCTION TCPM F TCP AXIS SPAT PATHCTRL AXIS	Tool tip moves along a straight line
14 FUNCTION TCPM F TCP AXIS POS PATHCTRL VECTOR	Tool tip and tool directional vector move in one plane
...	

Resetting the TCPM FUNCTION



- **FUNCTION RESET TCPM** is to be used if you want to purposely reset the function within a program.



The TNC automatically resets **TCPM FUNCTION** if you select a new program in a program run mode.
 You can reset the **TCPM FUNCTION** only if the **PLANE** function is inactive. If required, run **PLANE RESET** before **FUNCTION RESET TCPM**.

Example NC blocks

...	
25 FUNCTION RESETTCPM	Reset TCPM FUNCTION
...	

12.6 Three-dimensional tool compensation (Option #9)

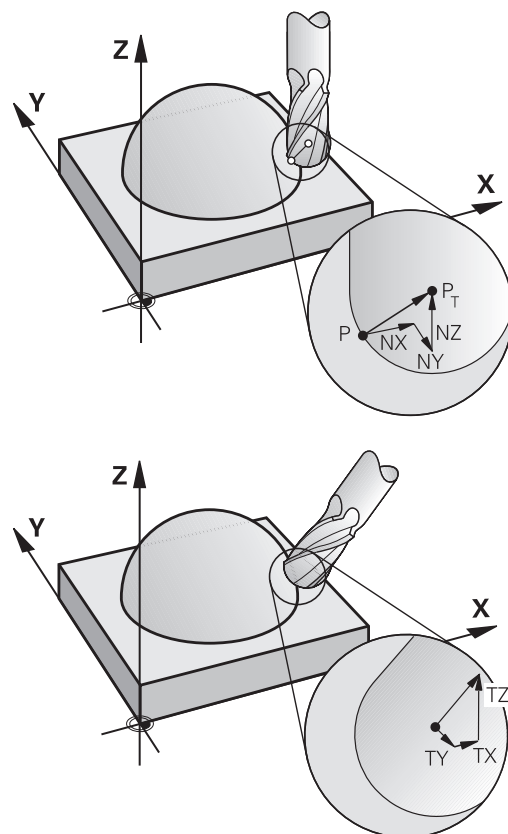
12.6 Three-dimensional tool compensation (Option #9)

Introduction

The TNC can carry out a three-dimensional tool compensation (3-D compensation) for straight-line blocks. Apart from the X, Y and Z coordinates of the straight-line end point, these blocks must also contain the components NX, NY and NZ of the surface-normal vector see "Definition of a normalized vector", page 475.

If you want to carry out a tool orientation, these blocks need also a normalized vector with the components TX, TY and TZ, which determines the tool orientation, see "Definition of a normalized vector", page 475.

The straight-line end point, the components for the surface-normal vector as well as those for the tool orientation must be calculated by a CAM system.



Application possibilities

- Use of tools with dimensions that do not correspond with the dimensions calculated by the CAM system (3-D compensation without definition of the tool orientation).
- Face milling: compensation of the cutter geometry in the direction of the surface-normal vector (3-D compensation with and without definition of the tool orientation). Cutting is usually with the end face of the tool.
- Peripheral milling: compensation of the cutter radius perpendicular to the direction of movement and perpendicular to the tool direction (3D radius compensation with definition of the tool orientation). Cutting is usually with the lateral surface of the tool.

Definition of a normalized vector

A normalized vector is a mathematical quantity with a value of 1 and any direction. The TNC requires up to two normalized vectors for LN blocks, one to determine the direction of the surface-normal vector, and another (optional) to determine the tool orientation direction. The direction of a surface-normal vector is determined by the components NX, NY and NZ. With an end mill and a radius mill, this direction is perpendicular from the workpiece surface to the tool datum PT, and with a toroid cutter through PT' or PT (see figure). The direction of the tool orientation is determined by the components TX, TY and TZ.



The coordinates for the X, Y, Z positions and the surface-normal components NX, NY, NZ, as well as TX, TY, TZ must be in the same sequence in the NC block.

Always indicate all of the coordinates and all of the surface-normal vectors in an LN block, even if the values have not changed from the previous block.

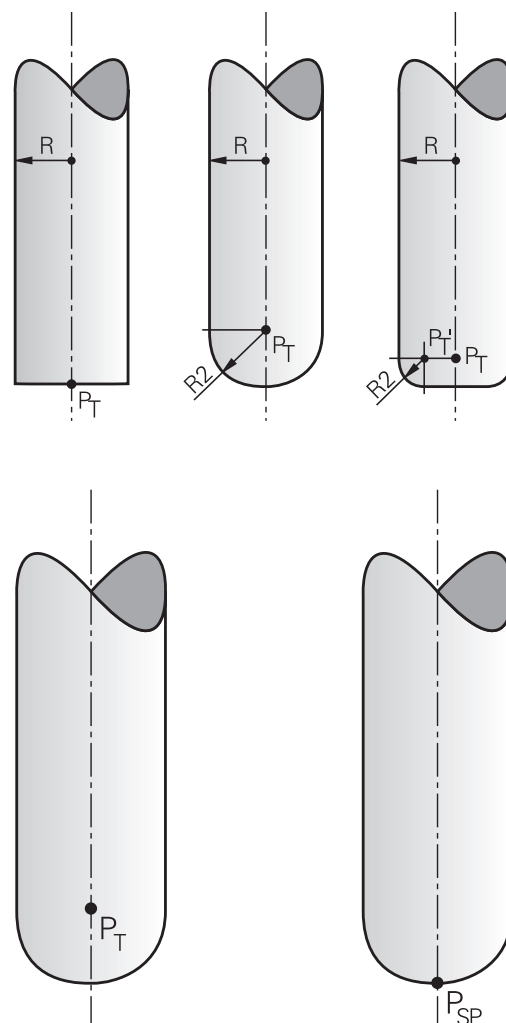
TX, TY and TZ must always be defined with numerical values. You cannot use Q parameters.

Calculate the normal vectors as exactly as possible and output them with a sufficient number of decimal places, in order to avoid interruptions in the feed rate during machining.

3-D compensation with surface-normal vectors is only effective for coordinates in the main axes X, Y, Z. If you insert a tool with oversize (positive delta value), the TNC outputs an error message. You can suppress the error message with the M function **M107** (see "Definition of a normalized vector", page 475).

The TNC will not display an error message if an entered tool oversize would cause damage to the contour.

Machine parameter **toolRefPoint** defines whether the CAD system has calculated the tool length compensation from the center of sphere PT or the south pole of the sphere PSP (see figure).



12.6 Three-dimensional tool compensation (Option #9)

Permissible tool shapes

You can describe the permissible tool shapes in the tool table via tool radii **R** and **R2** (see figure):

- Tool radius **R**: Distance from the tool center to the tool circumference
- Tool radius **R2**: Radius of the curvature between tool tip and tool circumference

The ratio of **R** to **R2** determines the shape of the tool:

- **R2** = 0: End mill
- **R2** = **R**: Radius cutter
- $0 < \mathbf{R2} < \mathbf{R}$: Toroid cutter

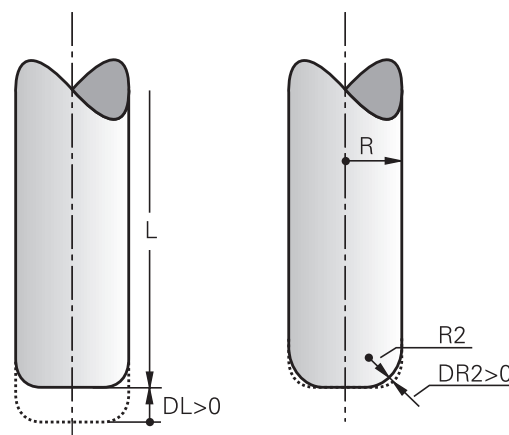
These data also provide the coordinates of the tool datum PT.

Using other tools: Delta values

If you want to use tools that have different dimensions than the ones you originally programmed, you can enter the difference between the tool lengths and radii as delta values in the tool table or **TOOL CALL**:

- Positive delta value **DL, DR, DR2**: The tool is larger than the original tool (oversize).
- Negative delta value **DL, DR, DR2**: The tool is smaller than the original tool (undersize).

The TNC then compensates the tool position by the sum of the delta values from the tool table and the tool call.



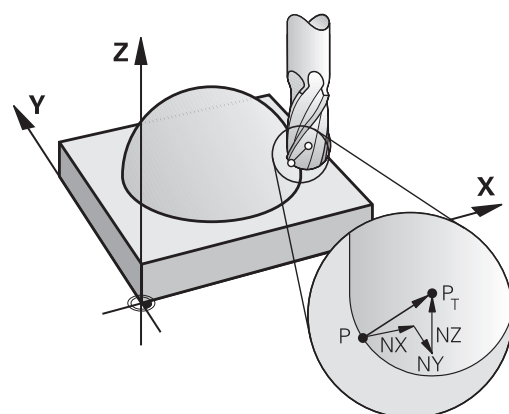
3-D compensation without TCPM

The TNC carries out a 3-D compensation for three-dimensional machining operations if the NC program contains surface-normal vectors. In this case, the **RL/RR** radius compensation and **TCPM** or **M128** must be inactive. The TNC displaces the tool in the direction of the surface-normal vectors by the sum of the delta values (tool table and **TOOL CALL**).

Example: Block format with surface-normal vectors

```
1 LN X+31.737 Y+21.954 Z+33.165NX+0.2637581 NY+0.0078922
  NZ-0.8764339 F1000 M3
```

LN:	Straight line with 3-D compensation
X, Y, Z:	Compensated coordinates of the straight-line end point
NX, NY, NZ:	Components of the surface-normal vector
F:	Feed rate
M:	Miscellaneous function



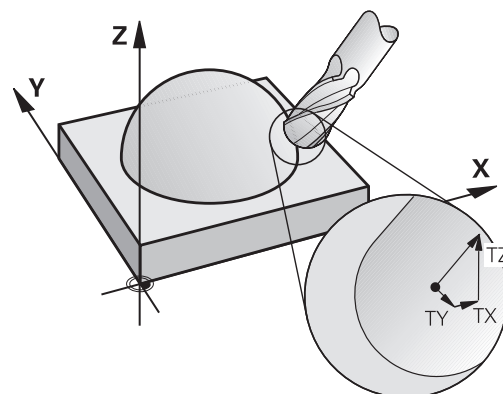
Three-dimensional tool compensation (Option #9) 12.6

Face Milling: 3D compensation with TCPM

Face milling is a machining operation carried out with the front face of the tool. A three-dimensional compensation is carried out during five-axis machining if the NC program contains surface-normal vectors and **TCPM** or **M128** is active. In this case, the RL/RR radius compensation must not be active. The TNC displaces the tool in the direction of the surface-normal vectors by the sum of the delta values (tool table and **TOOL CALL**).

If **TCPM** (see "Maintaining the position of the tool tip when positioning with tilted axes (TCPM): M128 (Option #9)", page 464) is active, the TNC maintains the tool perpendicular to the workpiece contour if no tool orientation is programmed in the **LN** block.

If there is a tool orientation **T** defined in the **LN** block and M128 (or **TCPM FUNCTION**) is active at the same time, then the TNC will position the rotary axes automatically so that the tool can reach the defined orientation. If you have not activated **M128** (or **TCPM FUNCTION**), then the TNC ignores the direction vector **T**, even if it is defined in the **LN** block.



The TNC is not able to automatically position the rotary axes on all machines. Refer to your machine manual.



Danger of collision!

On machines whose rotary axes only allow limited traverse, sometimes automatic positioning can require the table to be rotated by 180°. In this case, make sure that the tool head does not collide with the workpiece or the clamps.

12.6 Three-dimensional tool compensation (Option #9)

Example: Block format with surface normals without tool orientation

```
LN X+31.737 Y+21.954 Z+33.165 NX+0.2637581 NY+0.0078922
NZ-0.8764339 F1000 M128
```

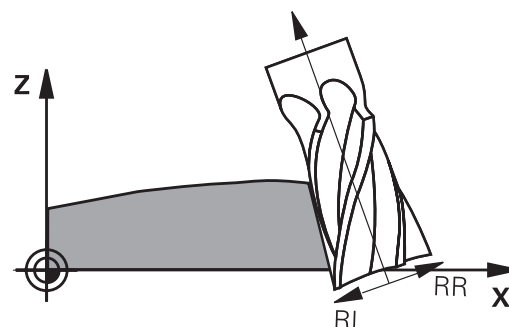
Example: Block format with surface normals and tool orientation

```
LN X+31.737 Y+21.954 Z+33.165 NX+0.2637581 NY+0.0078922
NZ-0.8764339 TX+0.0078922 TY-0.8764339 TZ+0.2590319
F1000 M128
```

LN:	Straight line with 3-D compensation
X, Y, Z:	Compensated coordinates of the straight-line end point
NX, NY, NZ:	Components of the surface-normal vector
TX, TY, TZ:	Components of the normalized vector for workpiece orientation
F:	Feed rate
M:	Miscellaneous function

Peripheral Milling: 3-D radius compensation with TCPM and radius compensation (RL/RR)

The TNC displaces the tool perpendicular to the direction of movement and perpendicular to the tool direction by the sum of the delta values **DR** (tool table and **TOOL CALL**). Determine the compensation direction with radius compensation **RL/RR** (see figure, traverse direction Y+). For the TNC to be able to reach the set tool orientation, you need to activate the function **M128**, see "Maintaining the position of the tool tip when positioning with tilted axes (TCPM): M128 (Option #9)", page 464. The TNC then positions the rotary axes automatically so that the tool can reach the defined orientation with the active compensation.



This function is possible only on machines for which you can define spatial angles for the tilting axis configuration. Refer to your machine manual.

The TNC is not able to automatically position the rotary axes on all machines.

Refer to your machine manual.

Note that the TNC makes a compensating movement by the defined **delta values**. The tool radius R defined in the tool table has no effect on the compensation.



Danger of collision!

On machines whose rotary axes only allow limited traverse, sometimes automatic positioning can require the table to be rotated by 180°. In this case, make sure that the tool head does not collide with the workpiece or the clamps.

Three-dimensional tool compensation (Option #9) 12.6

There are two ways to define the tool orientation:

- In an LN block with the components TX, TY and TZ
- In an L block by indicating the coordinates of the rotary axes

Example: Block format with tool orientation

```
1 LN X+31.737 Y+21.954 Z+33.165 TX+0.0078922 TY-0.8764339
  TZ+0.2590319 RR F1000 M128
```

LN:	Straight line with 3-D compensation
X, Y, Z:	Compensated coordinates of the straight-line end point
TX, TY, TZ:	Components of the normalized vector for workpiece orientation
RR:	Tool radius compensation
F:	Feed rate
M:	Miscellaneous function

Example: Block format with rotary axes

```
1 L X+31,737 Y+21,954 Z+33,165 B+12,357 C+5,896 RL F1000
  M128
```

L:	Straight line
X, Y, Z:	Compensated coordinates of the straight-line end point
B, C:	Coordinates of the rotary axes for tool orientation
RL:	Radius Compensation
F:	Feed rate
M:	Miscellaneous function

13

**Programming:
Pallet editor**

Programming: Pallet editor

13.1 Pallet management

13.1 Pallet management

Application



Pallet table management is a machine-dependent function. The standard functional range is described below. Refer to your machine manual.

Pallet tables (**.P**) are used for machining centers with pallet changers: The pallet table calls the part programs that are required for the different pallets, and activates presets, datum shifts and datum tables.

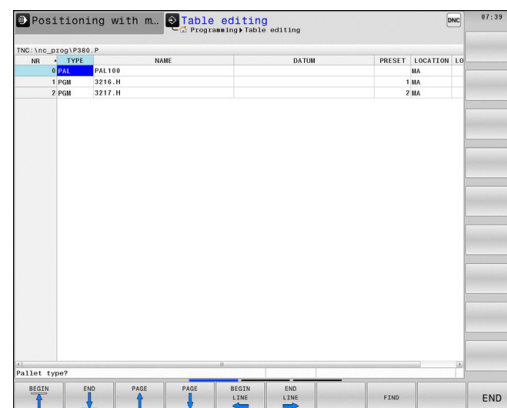
You can also use pallet tables to run in succession several programs that have different reference points.


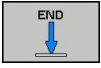


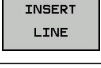
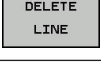
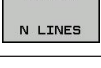
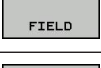
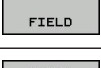
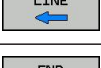
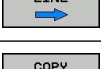
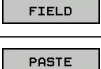
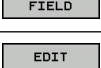
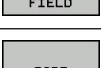




If you want to create or manage pallet tables, the name of the file must begin with a letter.

Pallet tables contain the following information:

- **TYPE** (entry obligatory): Identification for pallet or NC program (select with **ENT**)
- **NAME** (entry obligatory): Pallet or program name. The machine tool builder determines the pallet name (see your machine tool manual). The program name must be stored in the same directory as the pallet table. Otherwise you must enter the full path name for the program
- **PRESET** (entry optional): Preset number from the preset table. The preset number defined here is interpreted by the TNC as a workpiece datum.
- **DATUM** (entry optional): Name of the datum table. The datum table must be stored in the same directory as the pallet table. Otherwise you must enter the full path name for the datum table. Datums from the datum table can be activated in the NC program with Cycle 7 **DATUM SHIFT**
- **LOCATION** (entry obligatory): The entry "**MA**" indicates that the machine is loaded with a pallet or fixture that can be machined. The TNC only machines pallets or fixtures identified by "**MA**". To enter "**MA**", press the ENT key. Press the NO ENT key to remove the entry.
- **LOCK** (entry optional): Lock execution of a pallet line. Press the ENT key to mark the execution of a pallet line as locked (the affected line will be identified by "*****"). Press the NO ENT key to cancel the lock. You can lock the execution for individual programs, fixtures or entire pallets. Non-locked lines (e.g. PGM) of a locked pallet will also not be executed.



Soft key	Editing function
	Select beginning of table
	Select end of table
	Select previous page in table
	Select next page in table
	Insert as last line in the table
	Delete the last line in the table
	Add the number of lines that can be entered at the end of the table
	Copy highlighted field
	Insert copied field
	Select beginning of line
	Select end of line
	Copy the current value
	Insert the current value
	Edit the current field
	Sort by content of column
	Additional functions, e.g. saving

Programming: Pallet editor

13.1 Pallet management

Select pallet table

- ▶ Call the file manager in the Programming and Editing or Program Run mode: Press the **PGM MGT** key
- ▶ Display all type .P files: Press the **SELECT TYPE** and **SHOW ALL** soft keys
- ▶ Select a pallet table with the arrow keys, or enter a new file name to create a new table
- ▶ Confirm your entry with the **ENT** key

Exiting the pallet file

- ▶ To call the file manager, press the **PGM MGT** key
- ▶ To select a different type of file, press the **SELECT TYPE** soft key and the soft key for the desired file type, e.g. **SHOW .H**
- ▶ Select the desired file

Run pallet file



MP7683 defines whether the pallet table is to be executed blockwise or continuously.

Use the screen layout button to switch between table view and form view.

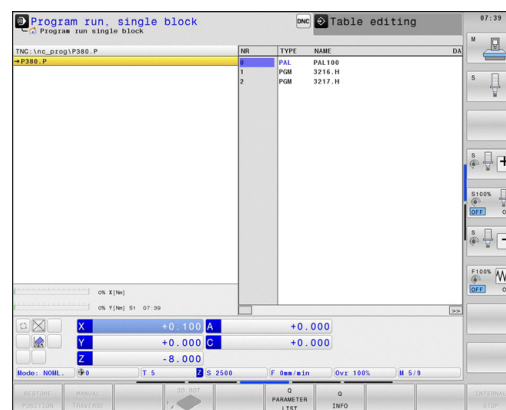
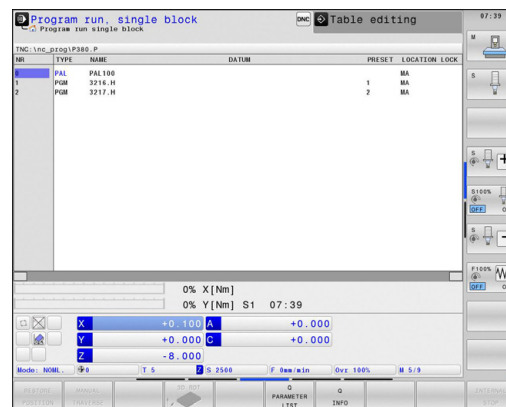
- ▶ Select the file manager in the Program Run, Full Sequence or Program Run, Single Block operating modes: Press the **PGM MGT** key
- ▶ Display all type .P files: Press the **SELECT TYPE** and **SHOW P.** soft keys
- ▶ Select the pallet table with the arrow keys and confirm with **ENT**
- ▶ Execute the pallet table: Press the NC start key

Pallet management 13.1

Screen layout for executing pallet tables

You can have the TNC display the program contents and pallet file contents on the screen together by selecting the screen layout **PROGRAM + PALLET**. During execution, the TNC then shows program blocks to the left and the pallet to the right. To check the program contents before execution, proceed as follows:

- ▶ Select a pallet table
- ▶ With the arrow keys, choose the program you would like to check
- ▶ Press the **OPEN PGM** soft key: the TNC displays the selected program on the screen. You can now page through the program with the arrow keys
- ▶ To return to the pallet table: Press the **OPEN PROGRAM** soft key again



14

**Programming:
Turning
Operations**

Programming: Turning Operations

14.1 Turning Operations on Milling Machines (Option #50)

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 operations are machining processes by which workpieces are rotated, thus implementing the cutting movements. 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, groove turning or thread turning.



The TNC offers you several cycles for each of the various production processes (see User's Manual, Cycles, "Turning" chapter).

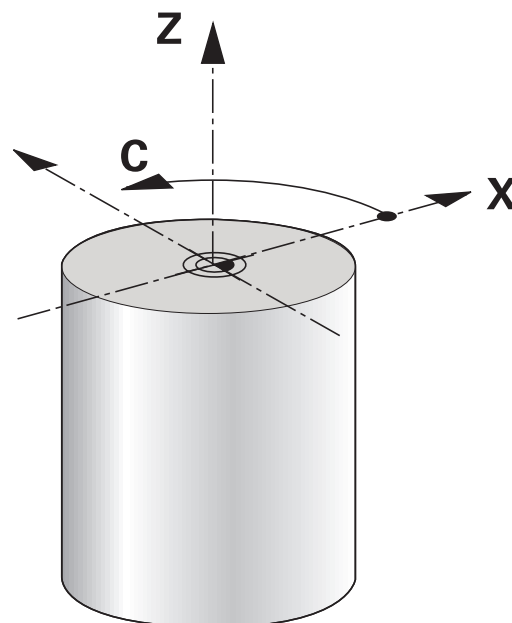
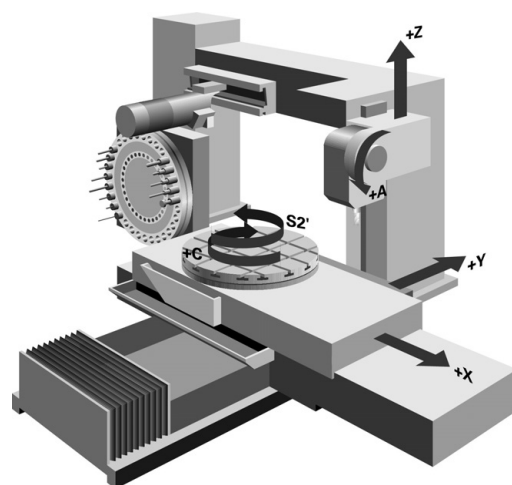
On the TNC you can simply switch between Milling and Turning mode within the NC program. In Turning mode, the rotary table serves as turning spindle, whereas the milling spindle with the tool is fixed. This enables rotationally symmetric contours to be created. The preset must be in the center of the turning spindle.

With the management of turning tools, other geometric descriptions are considered than with milling or drilling tools. To be able to execute tool radius compensation, for example, you have to define the tool radius. To support these definitions, the TNC provides a special tool management for turning tools, see "Tool data", page 499.

Different cycles are available for machining. These can also be used with additionally inclined swivel axes: see "Inclined turning", page 512

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 XZ coordinate plane. The machine axes to be used for the required motions 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.



14.2 Basic functions (Option 50)

Switching between milling/turning mode of operation



Switching the machine kinematics is a machine-dependent function.

The machine has to have been adapted by the machine manufacturer for turning operations and switching the mode of operation. Refer to your machine manual.

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 TNC shows a symbol in the status display when the turning mode is active

Icon	Mode of operation
	Turning mode active: FUNCTION MODE TURN
No symbol	Milling mode active: FUNCTION MODE MILL

When the modes of operation are switched between, the TNC executes a macro that defines the machine-specific settings for the specific mode of operation. 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.



The preset must be in the center of the turning spindle in turning mode.

The position of the tool tip must be aligned to the center of the turning spindle. Position the Y coordinates in Turning mode to the center of the turning spindle.

Check the orientation of the tool spindle. The tool tip must be aligned to the center of the turning spindle for outside machining. The tool tip must be aligned opposite to the center of the turning spindle for inside machining.

Check whether the rotation direction of the turning spindle is correct for the loaded tool.

If you process heavy workpieces with high speeds then high physical forces occur. Ensure that the workpiece is firmly clamped to avoid accidents or machine damage.

Programming: Turning Operations

14.2 Basic functions (Option 50)



In Turning mode, diameter values are displayed on the X axis position display. The TNC then shows a diameter symbol on the position display.

In Turning mode, the spindle potentiometer is effective on the turning spindle (rotary table).

Switching mode is not possible if "Tilting the working plane" or TCPM is active.

In Turning mode, no coordinate conversions are permitted except for the datum shift cycle.

You can also use the smartSelect function for defining the turning functions, see "Overview of special functions", page 390.

Entering the operation mode:

SPEC
FCT

- Show the soft-key row with special functions

TURNING
PROGRAM
FUNCTIONS

- Select the menu for **TURNING PROGRAM FUNCTIONS**

BASIC
FUNCTIONS

- Select **BASIC FUNCTIONS**

FUNCTION
MODE

- Select **FUNCTION MODE**

TURN

- Select the function for Turning or Milling mode
- Select the kinematic model that is to be activated with switchover (machine-dependent function). If you do not want to define a kinematic model, confirm with the NO ENT key

NC syntax

11 FUNCTION MODE TURN "AC_TABLE" ; ACTIVATE TURNING MODE

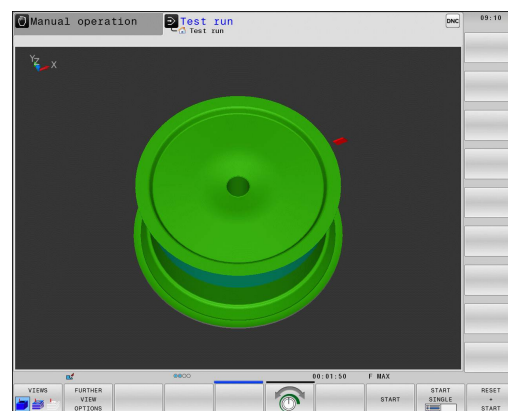
12 FUNCTION MODE MILL "B_HEAD" ; ACTIVATE MILLING MODE

Graphical display of turning operations

You can simulate turning operations in the **Test Run** mode. The requirement for this is a workpiece blank definition suitable for the turning process and Option #20.



The machining times shown for milling/turning operations in the simulation do not correspond to the actual machining times.

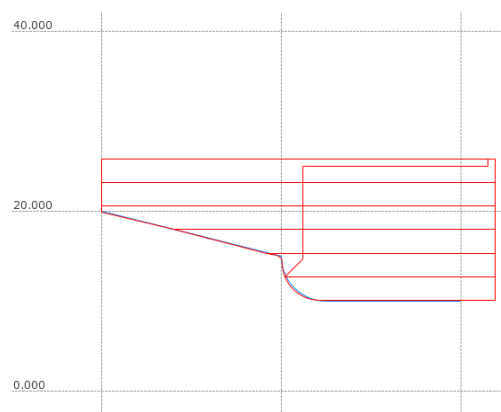


Graphic display in the Programming mode of operation

You can graphically simulate turning processes with the line graphic in **Programming** operating mode. To display the traverse movements in turning mode in the **Programming** operating mode, change the layout using the soft keys, see "Generating a graphic for an existing program", page 153.

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 when turning occurs in a 2D plane (X and Z coordinates), with a rectangular workpiece blank you must still program the Y values when defining the workpiece blank.



NC syntax

0 BEGIN PGM SHOULDER MM	
1 BLK FORM 0.1Y X+0 Y-1 Z-50	Workpiece blank definition
2 BLK FORM 0.2 X+87 Y+1 Z+2	
3 TOOL CALL 12	Tool call
4 M140 MB MAX	Retract the tool
5 FUNCTION MODE TURN	Activate Turning mode

Programming: Turning Operations

14.2 Basic functions (Option 50)

Program spindle speed



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 TNC modifies speed according to the distance of the tool tip to the center of the turning spindle. The TNC increases table speed with positioning in the direction of the turning center and reduces speed with movements away from the turning center.

For processing with constant spindle speed **VCONST:OFF**, speed is independent of the tool position.

Use **FUNCTION TURNDATA SPIN** to define the speed. The TNC now provides the following entry elements:

- **VCONST**: Constant cutting speed on/off (obligatory)
- **VC**: Cutting speed (optional)
- **S**: Nominal speed when no constant cutting speed is active (optional)
- **S MAX**: Maximum speed with constant cutting speed (optional), is reset with **S MAX 0**
- **gearrange**: Gear range for the turning spindle (optional)

Defining the speed:

SPEC FCT

TURNING PROGRAM FUNCTIONS

FUNCTION TURNDATA

TURNDATA SPIN

VCONST: ON

► Show the soft-key row with special functions

► Select the menu for **TURNING PROGRAM FUNCTIONS**

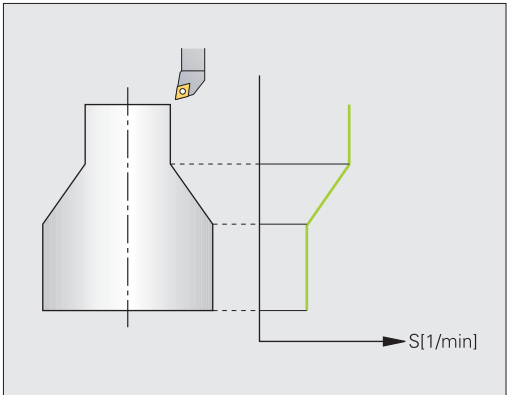
► Select **FUNCTION TURNDATA**

► Select **TURNDATA SPIN**

► Select the function for speed entry **VCONST:**



Cycle 800 limits maximum speed with eccentric turning. To reset, program **FUNCTION TURNDATA SPIN SMAX0**.



NC syntax

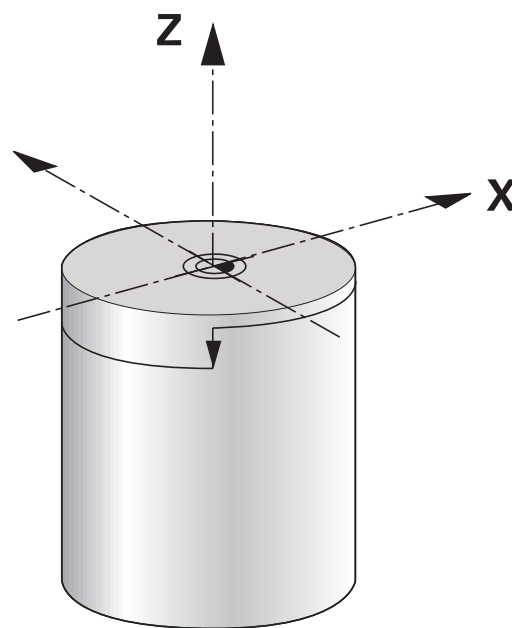
3 FUNCTION TURNDATA SPIN VCONST:ON VC:100 GEARRANGE:2	Definition of a constant cutting speed in gear range 2
3 FUNCTION TURNDATA SPIN VCONST:OFF S550	Definition of a constant spindle speed
...	

Feed rate

With turning, feed rates are often specified in millimeters per revolution. The TNC moves the tool according to a defined value for each spindle revolution. The resulting contouring feed rate is thus dependent on the speed of the turning spindle. With high speeds the TNC increases the feed rate and with low speeds reduces the feed rate. With uniform cutting depth you can machine with constant cutting force to achieve a constant cut thickness.

The programmed feed rate on a TNC is by default always interpreted in millimeters per minute (mm/min). If you wish to define feed rate in millimeters per revolution (mm/rev.), you must program **M136**. The TNC then interprets all subsequent feed rate specifications in mm/rev. until **M136** is canceled.

M136 is effective modally at the beginning of the block and can be canceled with **M137**.



NC syntax

10 L X+102 Z+2 R0 FMAX	Movement at rapid traverse
...	
15 L Z-10 F200	Movement at a feed rate of 200 mm/min
...	
19 M136	Feed rate in millimeters per revolution
20 L X+154 F0.2	Movement at a feed rate of 0.2 mm/rev.
...	

Programming: Turning Operations

14.3 Unbalance functions (Option #50)

14.3 Unbalance functions (Option #50)

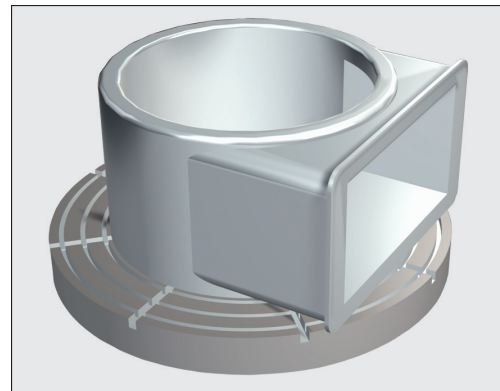
Unbalance while turning

General information



The machine has to have been adapted by the machine manufacturer for monitoring and measuring unbalance. The unbalance functions are not required some machine types. These functions might not be available on your machine. Refer to your machine manual.

The unbalance functions described here are basic functions that are set up and adapted to the machine by the machine manufacturer. The scope and effect of the described functions may therefore vary from machine to machine. The machine manufacturer may also provide different unbalance functions. Refer to your machine manual.



In a turning operation, the tool is in a fixed position, whereas the rotary table and the clamped workpiece rotate. Depending on the size of the workpiece, the mass that is set in rotation can be very large. As the workpiece rotates, it creates an outward centrifugal force.

The centrifugal force that occurs basically depends on the rotational speed, the mass and the unbalance of the workpiece. An unbalance occurs if an object with a rotationally nonsymmetrical mass distribution is set in rotation. The rotation of the mass object creates outward-directed centrifugal forces. If the rotating mass is evenly distributed, the centrifugal forces cancel each other out.

Unbalance is significantly influenced by the structural shape of the workpiece (e.g. nonsymmetrical pump body) and by the chucking equipment. As these factors frequently cannot be changed, you should compensate any existing unbalance by means of balancing weights. The TNC provides the **MEASURE UNBALANCE** cycle for this purpose. The cycle determines the existing unbalance and calculates the mass and position of the required balancing weight.



The rotation of the workpiece creates centrifugal forces that can cause vibration (resonance), depending on the unbalance. This vibration has a negative effect on the machining process and reduces the tool life. High centrifugal forces can damage the machine or push the workpiece out of the fixture.

Check the unbalance whenever you clamp a new workpiece. If required, use balancing weights to compensate any unbalance.

The removal of material during machining will change the mass distribution within the workpiece. This may also have an influence on workpiece unbalance. Therefore, unbalance checks should also be carried out between machining steps.

Keep in mind the mass and unbalance of the workpiece when choosing the speed. Do not use high speeds with heavy workpieces or high unbalance loads.

Unbalance Monitor function

The Unbalance Monitor function monitors the unbalance of a workpiece in Turning mode. If a maximum unbalance limit specified by the machine manufacturer is exceeded, the TNC issues an error message and initiates an emergency stop. In addition, you can further decrease the permissible unbalance limit by setting the machine parameter **limitUnbalanceUsr**. If this limit is exceeded, the TNC will display an error message, The table rotation is not stopped in this case. The TNC automatically activates the Unbalance Monitor function when you switch to Turning mode. The unbalance monitor is effective until you switch back to Milling mode.

Programming: Turning Operations

14.3 Unbalance functions (Option #50)

Measure Unbalance cycle

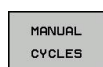
To ensure maximum safety and minimum strain on the machine and workpiece during turning, you should check the unbalance of the clamped workpiece and compensate it with a balancing weight. The TNC provides the **MEASURE UNBALANCE** cycle for this purpose.

The **MEASURE UNBALANCE** cycle determines the unbalance of the workpiece and calculates the mass and position of a balancing weight.

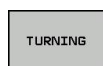
To determine unbalance:



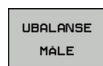
- ▶ Shift the soft-key row in the Manual Operation mode



- ▶ Select the **MANUAL CYCLES** soft key



- ▶ Select the **TURNING** soft key



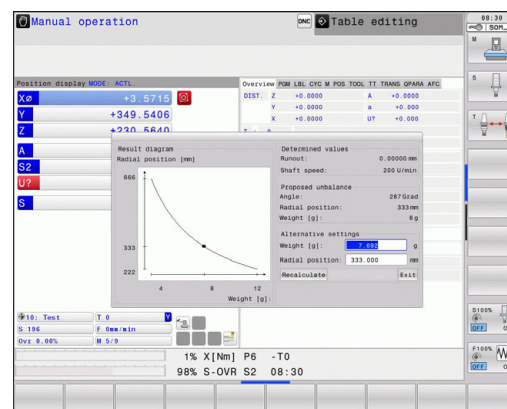
- ▶ Select the **MEASURE UNBALANCE** soft key
- ▶ Enter the speed for unbalance detection
- ▶ Press the NC Start button: The cycle starts rotating the table at a low speed and gradually increases the speed up to the defined value. The TNC displays a window that shows the calculated mass and radial position of the balancing weight.

If you wish to use a different radial position or mass for the balancing weight, you can overwrite one value and have the other value recalculated automatically.



Repeat the unbalance measurement after you have clamped a balancing weight.

In some cases, you may need to place two or more balancing weights at different positions in order to compensate unbalance.



14.4 Tools in turning mode (Option #50)

Tool call

Just as in Milling mode, turning tools are called with the **TOOL CALL** function. You merely have to enter the tool number or tool name in the **TOOL CALL** block.



You can call and insert a turning tool both in Milling mode and in Turning mode.

Tool selection in the pop-up window

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

In addition to the tool number and tool name, the control also shows the **ZL** and **XL** columns from the turning tool table.

NC syntax

1 FUNCTION MODE TURN	Turning mode selection
2 TOOL CALL "TRN_ROUGH"	Tool call
...	

Programming: Turning Operations

14.4 Tools in turning mode (Option #50)

Tool compensation in the 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.

FUNCTION TURNDATA CORR is always effective for the active tool. A renewed **TOOL CALL** deactivates compensation again. When you exit the program (e.g. PGM MGT), the TNC automatically resets the compensation values.

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

- **FUNCTION TURNDATA CORR-TCS**: The tool compensation is effective in the tool coordinate system
- **FUNCTION TURNDATA CORR-WCS**: 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.

Defining tool compensation:

SPEC
FCT

- ▶ Show the soft-key row with special functions

TURNING
PROGRAM
FUNCTIONS

- ▶ Select the menu for **TURNING PROGRAM FUNCTIONS**

FUNCTION
TURNDATA

- ▶ Select **FUNCTION TURNDATA**

TURNDATA
CORR

- ▶ Select **TURNDATA CORR**

NC syntax

```
21 FUNCTION TURNDATA CORR-TCS:Z/X DZL:0.1 DXL:0.05
```

```
...
```


Tools in turning mode (Option #50) 14.4

Tool data

You define turning-specific tool data in the turning tool table **TOOLTURN.TRN**.

The tool number saved in column **T** refers to the number of the turning tool in TOOL.T. Geometry values such as **L** and **R** from TOOL.T are not effective with turning tools.

In addition you must identify turning tools in the tool table TOOL.T as turning tools. For this, in column TYP select the tool type **TURN** for the appropriate tool. If you require additional geometric data for a tool you can create further indexed tools for this.



The tool number in TOOLTURN.TRN must match the tool number of the turning tool in TOOL.T. If you enter or copy a new line you can then enter the corresponding number.

Below the table window the TNC displays dialog text, unit specification and entry area for the specific input field

T	NAME	ZL	XL	YL	DZL	DXL
S0		75	10	0	0	0
S1		75	10	0	0	0
S2		70	0	0	0	0
S3		120	10	0	0	0

Other tool tables that are to be archived or used for test runs are given different file names with the extension .TRN.

Programming: Turning Operations

14.4 Tools in turning mode (Option #50)

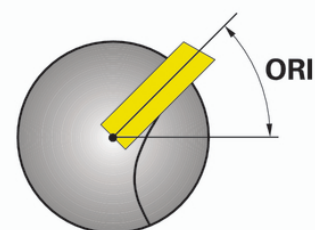
Tool data in the turning tool table

Input element	Application	Input
T	Tool number: Must match the tool number of the turning tool in TOOL.T	-
NAME	Tool name: The TNC automatically takes on the tool name if you select the turning tool table in the tool table	Input range: 32 characters max., only capital letters, no space characters
ZL	Compensation value for tool length 1 (Z direction)	-99999.9999...+99999.9999
XL	Compensation value for tool length 2 (X direction)	-99999.9999...+99999.9999
YL	Compensation value for tool length 3 (Y direction)	-99999.9999...+99999.9999
DZL	Delta value for tool length 1 (Z direction), additive effect on ZL	-99999.9999...+99999.9999
DXL	Delta value for tool length 2 (X direction), additive effect on XL	-99999.9999...+99999.9999
DYL	Delta value for tool length 3 (Y direction), additive effect on YL	-99999.9999...+99999.9999
RS	Tool tip radius: The TNC considers the tool tip radius in turning cycles and implements tool tip radius compensation when contours with radius compensation RL or RR were programmed	-99999.9999...+99999.9999
TO	Tool orientation: Direction of tool tip	1 to 9
ANGLE OF ORIENTATION (ORI)	Spindle orientation angle: Angle of the milling spindle for aligning the turning tool to the machining position	-360.0...+360.0
T-ANGLE	Setting angle for roughing and finishing tools	0.0000...+179.9999
P-ANGLE	Point angle for roughing and finishing tools	0.0000...+179.9999
CUTLENGTH	Cutting length of recessing tool	0.0000...+99999.9999
CUTWIDTH	Width of recessing tool	0.0000...+99999.9999
TYPE	Type of turning tool: Roughing tool ROUGH , finishing tool FINISH , thread tool THREAD , recessing tool RECESS , button tool BUTTON , groove turning tool RECTURN	ROUGH, FINISH, THREAD, RECESS, BUTTON, RECTURN

With the spindle orientation angle **ORI** you define the angle position of the milling spindle for the turning tool. Orient the tool tip depending on the tool orientation **TO** to the rotary table center or in the opposite direction.



The tool must be clamped and measured in the correct position.
Check the tool orientation after definition of a tool.

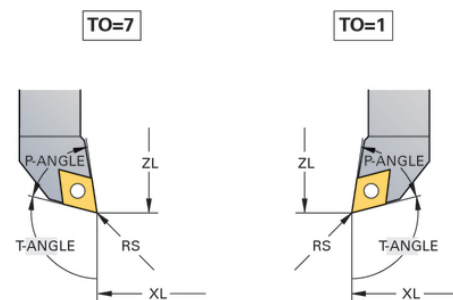


Tools in turning mode (Option #50) 14.4

Tool data for turning tool

Required and optional tool data for turning tool

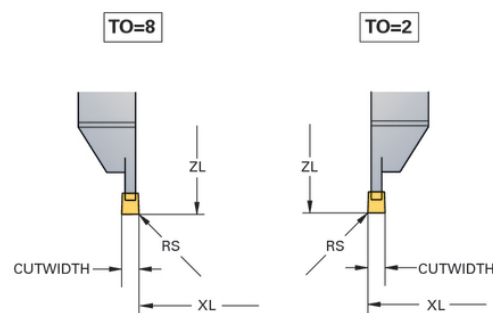
Input element	Application	Input
ZL	Tool length 1	Required
XL	Tool length 2	Required
XL	Tool length 3	Optional
DZL	Wear compensation ZL	Optional
DXL	Wear compensation XL	Optional
DYL	Wear compensation YL	Optional
RS	Cutting radius	Required
TO	Tool orientation	Required
ORI	Orientation angle	Required
T-ANGLE	Tool angle	Required
P-ANGLE	Point angle	Required
TYPE	Tool type	Required



Tool data for recessing tools

Required and optional tool data for recessing tools

Input element	Application	Input
ZL	Tool length 1	Required
XL	Tool length 2	Required
YL	Tool length 3	Optional
DZL	Wear compensation ZL	Optional
DXL	Wear compensation XL	Optional
DYL	Wear compensation YL	Optional
RS	Cutting radius	Required
TO	Tool orientation	Required
ORI	Orientation angle	Required
CUTWIDTH	Width of recessing tool	Required
TYPE	Tool type	Required



Programming: Turning Operations

14.4 Tools in turning mode (Option #50)

Tool data for groove turning tools

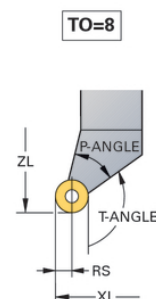
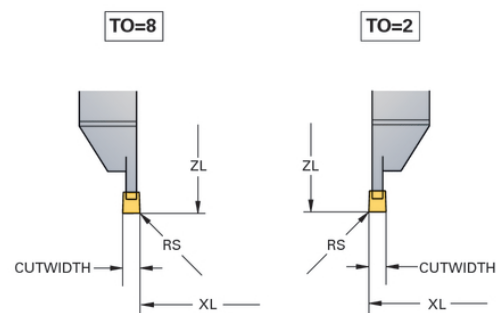
Required and optional tool data for groove turning tools

Input element	Application	Input
ZL	Tool length 1	Required
XL	Tool length 2	Required
YL	Tool length 3	Optional
DZL	Wear compensation ZL	Optional
DXL	Wear compensation XL	Optional
DYL	Wear compensation YL	Optional
RS	Cutting radius	Required
TO	Tool orientation	Required
ORI	Orientation angle	Required
CUTLENGTH	Cutting length of recessing tool	Required
CUTWIDTH	Width of recessing tool	Required
TYPE	Tool type	Required

Tool data for button tools

Required and optional tool data for button tools

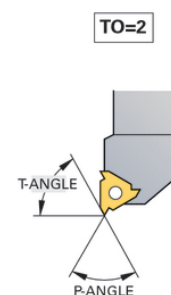
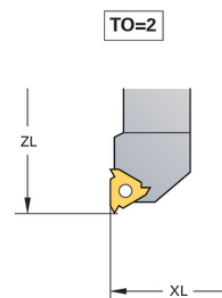
Input element	Application	Input
ZL	Tool length 1	Required
XL	Tool length 2	Required
YL	Tool length 3	Optional
DZL	Wear compensation ZL	Optional
DXL	Wear compensation XL	Optional
DYL	Wear compensation YL	Optional
RS	Cutting radius	Required
TO	Tool orientation	Required
ORI	Orientation angle	Required
T-ANGLE	Tool angle	Required
P-ANGLE	Point angle	Required
TYPE	Tool type	Required



Tool data for threading tools

Required and optional tool data for threading tools

Input element	Application	Input
ZL	Tool length 1	Required
XL	Tool length 2	Required
YL	Tool length 3	Optional
DZL	Wear compensation ZL	Optional
DXL	Wear compensation XL	Optional
DYL	Wear compensation YL	Optional
TO	Tool orientation	Required
ORI	Orientation angle	Required
T-ANGLE	Tool angle	Required
P-ANGLE	Point angle	Required
TYPE	Tool type	Required



Programming: Turning Operations

14.4 Tools in turning mode (Option #50)

Tool tip radius compensation TRC

Turning tools have a radius at the tool tip (**RS**). When machining tapers, chamfers and radii, this results in inaccuracies on the contour because programmed traverse paths are always referenced to the theoretical tool tip **S** (see figure at upper right). TRC prevents the resulting deviations.

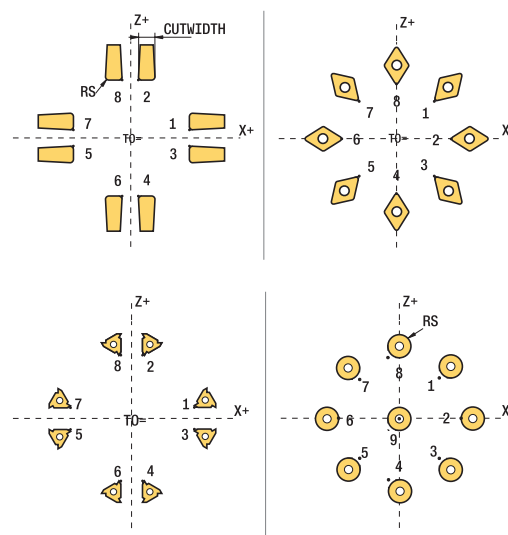
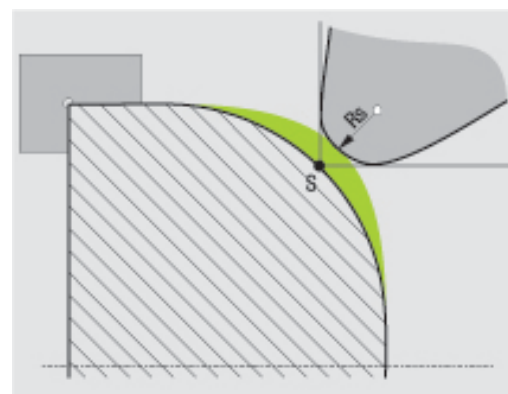
In turning cycles the TNC automatically carries out tool tip radius compensation. In specific traversing blocks and within programmed contours, activate TRC with **RL** or **RR**.

In turning cycles the TNC 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 TNC only as far as this is possible with the specific tool. The TNC outputs a warning when residual material is left behind.



When the position of the cutting edge is neutral (**TO=2;4;6;8**), the direction of the radius compensation is ambiguous. In this case, TRC is only possible within cycles.

The TNC can also run tool tip radius compensation during inclined processing. The following limitation applies here: if you activate inclined processing with **M128** then tool tip radius compensation without a cycle, i.e. in traversing blocks with **RL/RR**, is not possible. If you activate inclined processing with **M144** this limitation does not apply.



14.5 Turning program functions (Option #50)

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 complete recessing and undercutting as complete contour elements with a single NC block.



Recessing and undercutting always reference a previously defined linear contour element.
Recessing elements GRV and undercut elements UDC can only be used in contour subprograms that are called by a turning cycle (see User's Manual, Cycles, Turning).

You have various input possibilities when defining recessing and undercutting. Some of these inputs are mandatory, and others you can leave out (optional). The mandatory inputs are symbolized as such in the help graphics. In some elements you can select between two different definitions. The TNC has soft keys with the corresponding selection possibilities.

Programming recessing and undercutting:

-  ► Show the soft-key row with special functions
-  ► Select the menu for **TURNING PROGRAM FUNCTIONS**
-  ► Select **RECESS/ UNDERCUT**
-  ► Select **GRV** (recess) or **UDC** (undercut)

Programming: Turning Operations

14.5 Turning program functions (Option #50)

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 ends 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 elements in recessing GRV

Input element	Application	Input
CENTER	Center of recess	Required
R	Corner radius of both inner corners	Optional
DEPTH / DIAM	Recess depth (pay attention to the algebraic 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



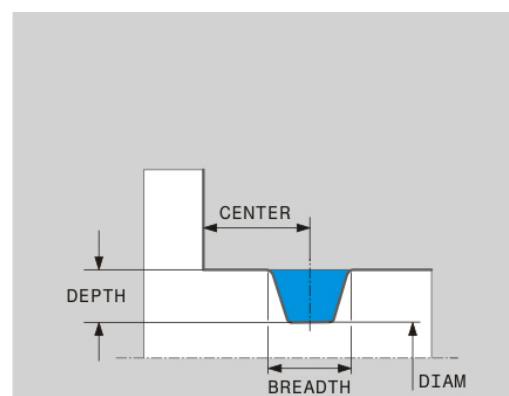
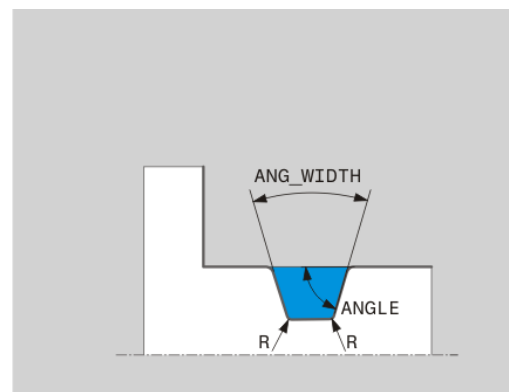
The algebraic sign for the recess depth specifies the machining position of the recess (inside/outside machining).

Algebraic sign of recess depth for outside machining:

- Use a negative sign when the contour element runs in a negative direction to the Z coordinate
- Use a positive sign when the contour element runs in a positive direction to the Z coordinate

Algebraic sign of recess depth for inside machining:

- Use a positive sign when the contour element runs in a negative direction to the Z coordinate
- Use a negative sign when the contour element runs in a positive direction to the Z coordinate



Radial recess: depth=5, width=10, Pos.= Z-15

21 L X+40 Z+0

22 L Z-30

23 GRV RADIAL CENTER-15 DEPTH-5 BREADTH10 CHF1 FAR_CHF1

24 L X+60

Programming undercutting

Undercutting is usually required for the flush connection of counterparts. In addition undercutting can help to reduce the notch effect at corners. Threads and fits are often machined with an undercut. You have various contour elements for defining the different undercuts:

- **UDC TYPE_E**: Undercut for cylindrical surface to be further processed in compliance with DIN 509
- **UDC TYPE_F**: Undercut for plan and cylindrical surface for further processing in compliance with DIN 509
- **UDC TYPE_H**: Undercut for more rounded transition in compliance with DIN 509
- **UDC TYPE_K**: Undercut in face and cylindrical surface
- **UDC TYPE_U**: Undercut in cylindrical surface
- **UDC THREAD**: Thread undercut in compliance with DIN 76



The TNC always interprets undercuts as form elements in the longitudinal direction. No undercuts are possible in the plane direction.

Programming: Turning Operations

14.5 Turning program functions (Option #50)

Undercut DIN 509 UDC TYPE_E

Input elements in undercut DIN 509 UDC TYPE_E

Input element	Application	Input
R	Corner radius of both inner corners	Optional
DEPTH	Undercut depth	Optional
BREADTH	Undercut width	Optional
ANGLE	Undercut angle	Optional

Undercut: depth = 2, width = 15

21 L X+40 Z+0
22 L Z-30
23 UDC TYPE_E R1 DEPTH2 BREADTH15
24 L X+60

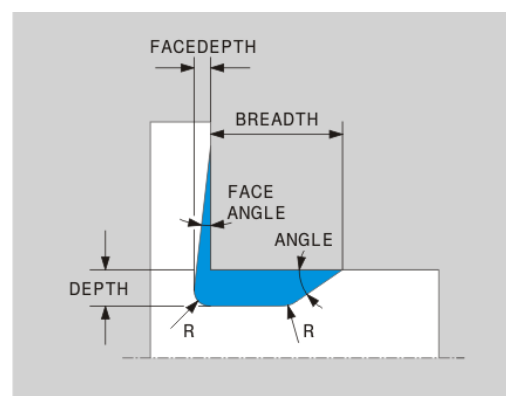
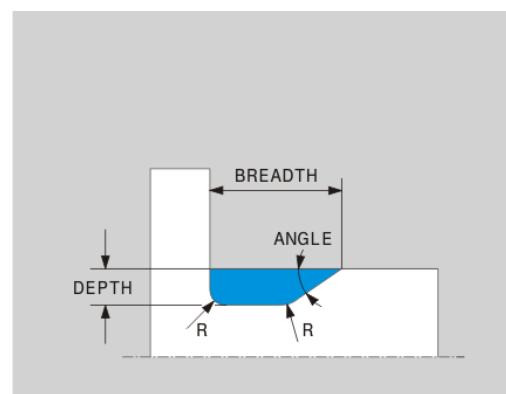
Undercut DIN 509 UDC TYPE_F

Input elements in undercut DIN 509 UDC TYPE_F

Input element	Application	Input
R	Corner radius of both inner corners	Optional
DEPTH	Undercut depth	Optional
BREADTH	Undercut width	Optional
ANGLE	Undercut angle	Optional
FACEDEPTH	Depth of face	Optional
FACEANGLE	Contour angle of face	Optional

Undercut form F: depth = 2, width = 15, depth of face = 1

21 L X+40 Z+0
22 L Z-30
23 UDC TYPE_F R1 DEPTH2 BREADTH15 FACEDEPTH1
24 L X+60

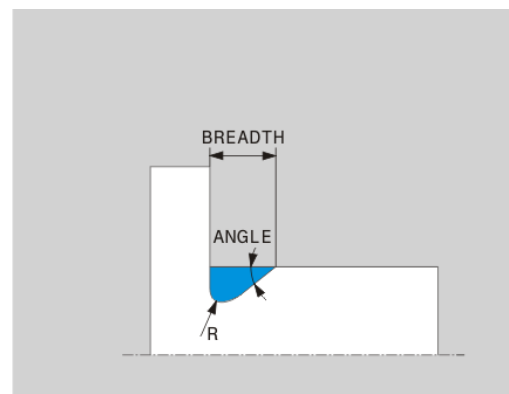


Undercut DIN 509 UDC TYPE_H**Input elements in undercut DIN 509 UDC TYPE_H**

Input element	Application	Input
R	Corner radius of both inner corners	Required
BREADTH	Undercut width	Required
ANGLE	Undercut angle	Required

Undercut form H: depth = 2, width = 15, angle = 10°

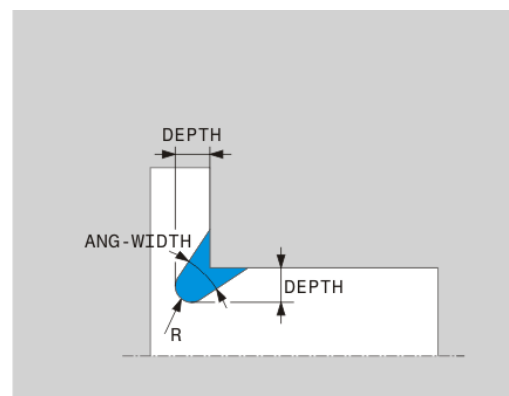
21 L X+40 Z+0
22 L Z-30
23 UDC TYPE_H R1 BREADTH10 ANGLE10
24 L X+60

**Undercut UDC TYPE_K****Input elements in undercut UDC TYPE_K**

Input element	Application	Input
R	Corner radius of both inner corners	Required
DEPTH	Undercut depth (paraxially)	Required
ROT	Angle to longitudinal axis (default: 45°)	Optional
ANG_WIDTH	Opening angle of undercut	Required

Undercut form K: depth = 2, width = 15, opening angle = 30°

21 L X+40 Z+0
22 L Z-30
23 UDC TYPE_K R1 DEPTH3 ANG_WIDTH30
24 L X+60



Programming: Turning Operations

14.5 Turning program functions (Option #50)

Undercut UDC TYPE_U

Input elements in undercut UDC TYPE_U

Input element	Application	Input
R	Corner radius of both inner corners	Required
DEPTH	Undercut depth	Required
BREADTH	Undercut width	Required
RND / CHF	Curve / chamfer of outer corner	Required

Undercut form U: depth = 3, width = 8

```

21 L X+40 Z+0
22 L Z-30
23 UDC TYPE_U R1 DEPTH3 BREADTH8 RND1
24 L X+60

```

Undercut UDC THREAD

Input elements in undercut DIN 76 UDC THREAD

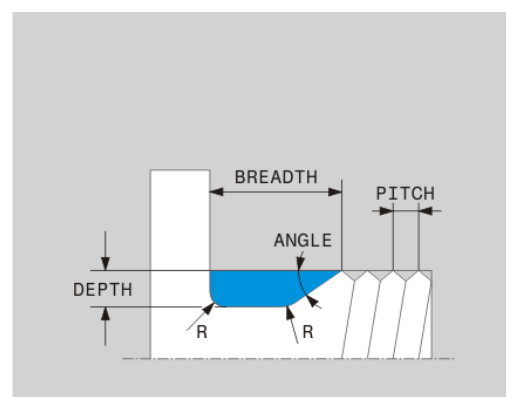
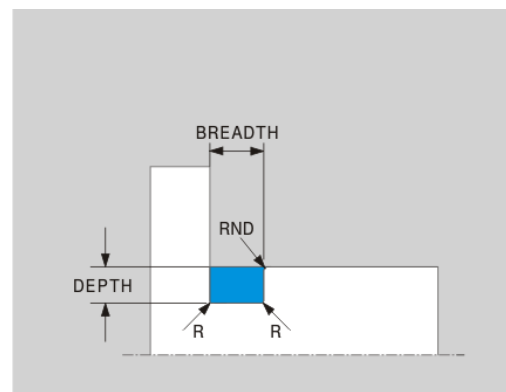
Input element	Application	Input
PITCH	Thread pitch	Optional
R	Corner radius of both inner corners	Optional
DEPTH	Undercut depth	Optional
BREADTH	Undercut width	Optional
ANGLE	Undercut angle	Optional

Thread undercut according to DIN 76: Thread pitch = 2

```

21 L X+40 Z+0
22 L Z-30
23 UDC THREAD PITCH2
24 L X+60

```







Blank form update TURNDATA BLANK






The **TURNDATA BLANK** function enables you to use the blank form update feature. The control detects the described contour and only then machines the residual material.

With **TURNDATA BLANK** you call a contour description used by the TNC as an updated workpiece blank.

Define the function TURNDATA BLANK as follows:






-  ► Show the soft-key row with special functions
-  ► Select the menu for **TURNING PROGRAM FUNCTIONS**
-  ► Select **FUNCTION TURNDATA**
-  ► Select **TURNDATA BLANK**
- Press the soft key for the desired contour call

You can call the contour description in the following ways:

Soft key	Call
	Contour description in an external program Call via file name
	Contour description in an external program Call via string parameter
	Contour description in a subprogram Call via label number
	Contour description in a subprogram Call via label name
	Contour description in a subprogram Call via string parameter

Deactivate blank form update

Deactivate blank form update as follows:

-  ► Show the soft-key row with special functions
-  ► Select the menu for **TURNING PROGRAM FUNCTIONS**
-  ► Select **FUNCTION TURNDATA**
-  ► Select **TURNDATA BLANK**
-  ► Select **BLANK OFF**

Inclined turning

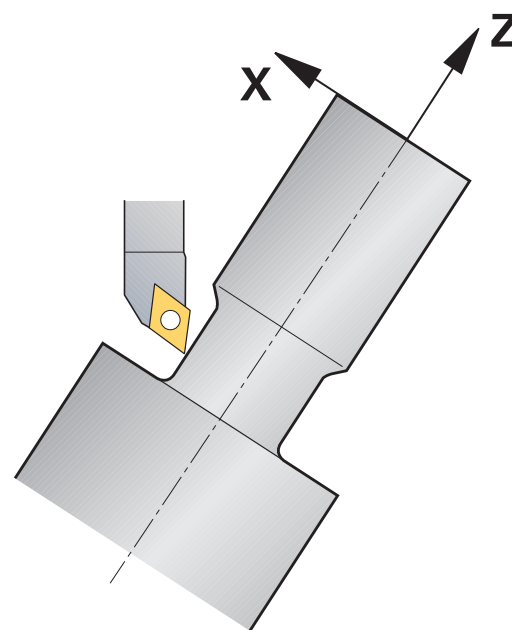
It may sometimes be necessary for you to bring the swivel axes into a specific position to machine a specific process. This can be necessary for example when you can only machine contour elements according to a specific position due to tool geometry.

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, so that the workpiece is sloping, the TNC runs traverse movements in the displaced workpiece coordinate system. If the inclined axis is a swivel head (tool is sloping) the workpiece coordinate system is not displaced.

After inclining the swivel axis you may have to again pre-position the tool in the Y coordinates and orient the position of the tool tip with the cycle 800.

Alternatively to function **M144** you can also use function **M128**. The effect is identical, but the following limitation applies: The TNC can also run tool tip radius compensation during inclined processing. If you activate inclined processing with **M128** then tool tip radius compensation without a cycle, i.e. in traversing blocks with **RL/RR**, is not possible. If you activate inclined processing with **M144** this limitation does not apply.

If the turning cycles are executed with **M144**, the angles of the tool to the contour change. The TNC automatically takes these modifications into account and thus also monitors the machining in inclined state.



You can use recessing cycles and thread cycles with inclined machining only with a rectangular tool angle (+90°, -90°).

Tool compensation **FUNCTION TURNDATA CORR-TCS** is always effective in the tool coordinate system, even during inclined machining.

Turning program functions (Option #50) 14.5

...	
12 M144	Activate inclined machining
13 L A-25 R0 FMAX	Position swivel axis
14 CYCL DEF 800 ADJUST XZ SYSTEM	Workpiece coordinate system and align tool
Q497=+90 ;PRECESSION ANGLE	
Q498=+0 ;REVERSE TOOL	
Q530=+2 ;INCLINED MACHINING	
Q531=-25 ;ANGLE OF INCIDENCE?	
Q532=750 ;FEED RATE	
Q533=+1 ;PREFERRED DIRECTION	
Q535=3 ;ECCENTRIC TURNING	
Q536=0 ;ECCENTRIC W/O STOP	
15 L X+165 Y+0 R0 FMAX	Pre-positioning the tool
16 L Z+2 R0 FMAX	Tool at starting position
...	Machining with inclined axis

15

**Manual operation
and setup**

Manual operation and setup

15.1 Switch-on, switch-off

15.1 Switch-on, switch-off

Switch-on



Switch-on and crossing over the reference points can vary depending on the machine tool.
Refer to your machine manual.

Switch on the power supply for TNC and machine. The TNC then displays the following dialog:

SYSTEM STARTUP

- ▶ TNC is started

POWER INTERRUPTED



- ▶ TNC message that the power was interrupted—clear the message

COMPILE A PLC PROGRAM

- ▶ The PLC program of the TNC is automatically compiled

RELAY EXT. DC VOLTAGE MISSING



- ▶ Switch on external dc voltage. The TNC checks the functioning of the EMERGENCY STOP circuit

MANUAL OPERATION

TRAVERSE REFERENCE POINTS



- ▶ Cross the reference points manually in the displayed sequence: For each axis press the machine START button, or



- ▶ Cross the reference points in any sequence: Press and hold the machine axis direction button for each axis until the reference point has been traversed



If your machine is equipped with absolute encoders, you can leave out crossing the reference marks. In such a case, the TNC is ready for operation immediately after the machine control voltage is switched on.

The TNC is now ready for operation in the **Manual Operation** mode.



The reference points need only be crossed if the machine axes are to be moved. If you intend only to write, edit or test programs, you can select the **Programming** or **Test Run** mode of operation immediately after switching on the control voltage. You can cross the reference points later by pressing the **PASS OVER REFERENCE** soft key in the **MANUAL OPERATION** mode.

Crossing the reference point in a tilted working plane



Danger of collision!

Make sure that the angle values entered in the menu for tilting the working plane match the actual angles of the tilted axis.

Deactivate the "Tilt Working Plane" function before you cross the reference points. Take care that there is no collision. Retract the tool from the current position first, if necessary.

The TNC automatically activates the tilted working plane if this function was enabled when the control was switched off. Then the TNC moves the axes in the tilted coordinate system when an axis-direction key is pressed. Position the tool in such a way that a collision is excluded during the subsequent crossing of the reference points. To cross the reference points you have to deactivate the "Tilt Working Plane" function, see "To activate manual tilting:", page 574.



If you use this function, then for non-absolute encoders you must confirm the positions of the rotary axes, which the TNC displays in a pop-up window. The position displayed is the last active position of the rotary axes before switch-off.

If one of the two functions that were active before is active now, the **NC START** button has no function. The TNC outputs a corresponding error message.

Manual operation and setup

15.1 Switch-on, switch-off

Switch-off



Deactivation is a machine-dependent function.
Refer to your machine manual.

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

- ▶ Select the **Manual Operation** mode



- ▶ Select the function for shutting down



- ▶ Confirm with the **SHUT DOWN** soft key
- ▶ When the TNC displays the message **Now you can switch off the TNC** in a pop-up window, you may cut off the power supply to the TNC



Caution: Data may be lost!

Inappropriate switch-off of the TNC can lead to data loss!

The control restarts after pressing the **RESTART** soft key. Switch-off during a restart can also result in data loss!

15.2 Moving the machine axes

Note



Traversing with the machine axis direction buttons can vary depending on the machine tool. Refer to your machine manual.

Moving the axis with the machine axis direction buttons



- Select the **Manual Operation** mode



- Press the machine axis direction button and hold it as long as you wish the axis to move, or



- Move the axis continuously: Press and hold the machine axis direction button, then press the machine START button



- Stop the axis: Press the machine STOP button

You can move several axes at a time with these two methods. The control then shows the feed rate. You can change the feed rate at which the axes are traversed with the **F** soft key, see "Spindle speed S, feed rate F and miscellaneous function M", page 530.

If a traverse task is active on the machine, the control displays the control-in-operation symbol.

Incremental jog positioning

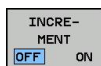
With incremental jog positioning you can move a machine axis by a preset distance.



- Select the **Manual Operation** or **El. Handwheel** mode of operation



- Shift the soft-key row



- Select incremental jog positioning: Switch the **INCREMENT** soft key to ON

JOG INCREMENT =



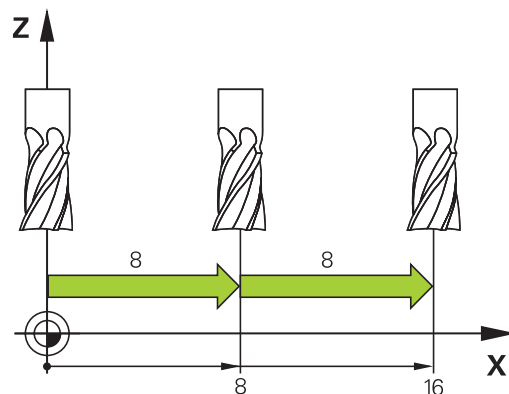
- Enter the jog increment in mm, and confirm with the **ENT** key



- Press the machine axis direction button as often as desired



The maximum permissible value for infeed is 10 mm.



Manual operation and setup

15.2 Moving the machine axes

Traverse with electronic handwheels

The TNC supports traversing with the following new electronic handwheels:

- HR 520: Handwheel compatible for connection to HR 420 with display, data transfer per cable
- HR 550 FS: Handwheel with display, radio data transmission

In addition to this, the TNC continues to support the cable handwheels HR 410 (without display) and HR 420 (with display).



Caution: Danger to the operator and handwheel!

All of the handwheel connectors may only be removed by authorized service personnel, even if it is possible without any tools!

Ensure that the handwheel is plugged in before you switch on the machine!

If you wish to operate your machine without the handwheel, disconnect the cable from the machine and secure the open socket with a cap!



Your machine tool builder can make additional functions of the HR 5xx available. Refer to your machine manual.



A HR 5xx handwheel is recommended if you want to use the handwheel superimposition in virtual axis function see "Virtual tool axis VT".

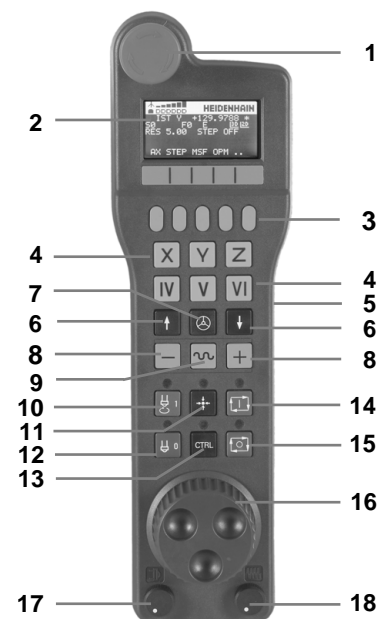
The portable HR 5xx handwheels feature a display on which the TNC shows information. In addition, you can use the handwheel soft keys for important setup functions, e.g. datum setting or entering and running M functions.

As soon as you have activated the handwheel with the handwheel activation key, the operating panel is locked. This is indicated by a pop-up window on the TNC screen.



Moving the machine axes 15.2

- 1** EMERGENCY STOP button
- 2** Handwheel display for status display and function selection; for further information, see: ""
- 3** Soft keys
- 4** Axis selection keys; can be exchanged by the machine manufacturer depending on the axis configuration
- 5** Permissive key
- 6** Arrow keys for defining handwheel sensitivity
- 7** Handwheel activation key
- 8** Key for TNC traverse direction of the selected axis
- 9** Rapid traverse superimposition for direction key
- 10** Spindle switch-on (machine-dependent function, key can be exchanged by the machine manufacturer)
- 11** "Generate NC block" key (machine-dependent function, key can be exchanged by the machine manufacturer)
- 12** Spindle switch-off (machine-dependent function, key can be exchanged by the machine manufacturer)
- 13** CTRL key for special functions (machine-dependent function, key can be exchanged by the machine manufacturer)
- 14** NC start (machine-dependent function, key can be exchanged by the machine manufacturer)
- 15** NC stop (machine-dependent function, key can be exchanged by the machine manufacturer)
- 16** Handwheel
- 17** Spindle speed potentiometer
- 18** Feed rate potentiometer
- 19** Cable connection, not available with the HR 550 FS wireless handwheel

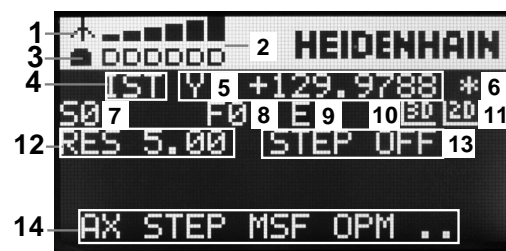


Manual operation and setup

15.2 Moving the machine axes

Handwheel display

- 1 Only with wireless handwheel HR 550 FS:** Shows whether the handwheel is in the docking station or whether wireless operation is active
- 2 Only with wireless handwheel HR 550 FS:** Shows the field strength, 6 bars = maximum field strength
- 3 Only with wireless handwheel HR 550 FS:** Shows the charge status of the rechargeable battery, 6 bars = fully charged A bar moves from the left to the right during recharging
- 4 ACTL:** Type of position display
- 5 Y+129.9788:** Position of the selected axis
- 6 *:** STIB (control in operation); program run has been started or axis is in motion
- 7 S0:::** Current spindle speed
- 8 F0:** Feed rate at which the selected axis is moving
- 9 E:** Error message
- 10 3D:** Tilted-working-plane function is active
- 11 2D:** Basic rotation function is active
- 12 RES 5.0:** Active handwheel resolution. Distance in mm/rev (°/rev for rotary axes) that the selected axis moves for one handwheel revolution
- 13 STEP ON or OFF:** Incremental jog active or inactive. If the function is active, the TNC also displays the active jog increment
- 14** Soft-key row: Selection of various functions, described in the following sections



Special features of the HR 550 FS wireless handwheel



Due to various potential sources of interference, a wireless connection is not as reliable as a cable connection. Before you use the wireless handwheel it must therefore be checked whether there are any other radio users in the surroundings of the machine. This inspection for presence of radio frequencies or channels is recommended for all industrial radio systems.

When the HR550 is not needed, always put it in the handwheel holder. This way you can ensure that the handwheel batteries are always ready for use thanks to the contact strip on the rear side of the wireless handwheel and the recharge control, and that there is a direct contact connection for the emergency stop circuit.

If an error (interruption of the radio connection, poor reception quality, defective handwheel component) occurs, the handwheel always reacts with an emergency stop.

Please read the notes on the configuration of the HR 550 FS wireless handwheel see "Configure HR 550 FS wireless handwheel", page 640



Caution: Danger to the operator and machine!

Due to safety reasons you must switch off the wireless handwheel and the handwheel holder after an operating time of 120 hours at the latest so that the TNC can run a functional test when it is restarted!

If you use several machines with wireless handwheels in your workshop you have to mark the handwheels and holders that belong together so that their respective associations are clearly identifiable (e.g. by color stickers or numbers). The markings on the wireless handwheel and the handwheel holder must be clearly visible to the user!

Before every use, make sure that the correct handwheel for your machine is active.



Manual operation and setup

15.2 Moving the machine axes

The HR 550 FS wireless handwheel features a rechargeable battery. The battery is recharged when you put the handwheel in the holder (see figure).

You can operate the HR 550 FS with the accumulator for up to 8 hours before it must be recharged again. It is recommended, however, that you always put the handwheel in its holder when you are not using it.

As soon as the handwheel is in its holder, it switches internally to cable operation. In this way you can use the handwheel even if it were completely discharged. The functions are the same as with wireless operation.



When the handwheel is completely discharged, it takes about 3 hours until it is fully recharged in the handwheel holder.

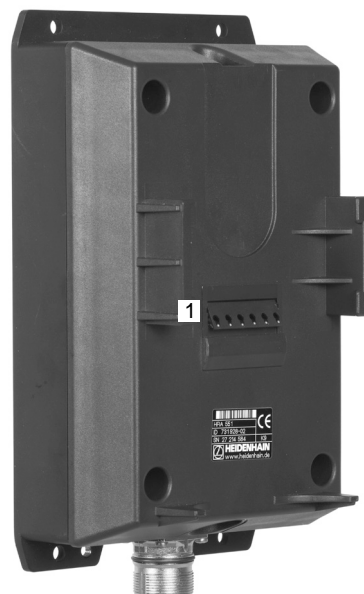
Clean the contacts **1** in the handwheel holder and of the handwheel regularly to ensure their proper functioning.

The transmission range is amply dimensioned. If you should nevertheless happen to come near the edge of the transmission area, which is possible with very large machines, the HR 550 FS warns you in time with a plainly noticeable vibration alarm. If this happens you must reduce the distance to the handwheel holder, into which the radio receiver is integrated.



Caution: Danger to the workpiece and tool!

If interruption-free operation is no longer possible within the transmission range the TNC automatically triggers an emergency stop. This can also happen during machining. Try to stay as close as possible to the handwheel holder and put the handwheel in its holder when you are not using it.



If the TNC has triggered an emergency stop you must reactivate the handwheel. Proceed as follows:

- ▶ Select the Programming and Editing mode of operation
- ▶ Select the MOD function: Press the MOD key
- ▶ Scroll through the soft-key row



- ▶ Select the configuration menu for the wireless handwheel: Press the **SET UP WIRELESS HANDWHEEL** soft key
- ▶ Click the **Start handwheel** button to reactivate the wireless handwheel
- ▶ To save the configuration and exit the configuration menu, press the **END** button

The MOD mode of operation includes a function for initial operation and configuration of the handwheel see "Configure HR 550 FS wireless handwheel", page 640.

Selecting the axis to be moved

You can activate directly through the axis address keys the principal axes X, Y, Z and three other axes defined by the machine tool builder. Your machine tool builder can also place the virtual axis VT directly on one of the free axis keys. If the virtual axis VT is not on one of the axis selection keys, proceed as follows:

- ▶ Press the handwheel soft key F1 (**AX**): The TNC displays all active axes on the handwheel display. The currently active axis blinks
- ▶ Select the desired axis with the handwheel soft keys F1 (->) or F2 (<-) and confirm with the handwheel soft key F3 (**OK**)

Setting the handwheel sensitivity

The handwheel sensitivity specifies the distance an axis moves per handwheel revolution. The sensitivity levels are pre-defined and are selectable with the handwheel arrow keys (only when incremental jog is not active).

Selectable sensitivity levels: 0.01/0.02/0.05/0.1/0.2/0.5/1/2/5/10/20 [mm/revolution or degrees/revolution]

Manual operation and setup

15.2 Moving the machine axes

Moving the axes



- ▶ To activate the handwheel, press the handwheel button on the HR 5xx: You can now only operate the TNC via the HR 5xx, and the TNC displays a pop-up window with text on the TNC screen
- ▶ Select the desired operating mode via the OPM soft key if necessary



- ▶ If required, press and hold the permissive button



- ▶ Use the handwheel to select the axis to be moved. Select the additional axes via soft key, if required



- ▶ Move the active axis in the positive direction, or



- ▶ Move the active axis in the negative direction



- ▶ To deactivate the handwheel, press the handwheel key on the HR 5xx: Now you can operate the TNC again via the operating panel

Potentiometer settings

The potentiometers of the machine operating panel continue to be active after you have activated the handwheel. If you want to use the potentiometers on the handwheel, proceed as follows:

- ▶ Press the **CTRL** and Handwheel keys on the HR 5xx. The TNC shows the soft-key menu for selecting the potentiometers on the handwheel display
- ▶ Press the **HW** soft key to activate the handwheel potentiometers

If you have activated the potentiometers on the handwheel, you must reactivate the potentiometers of the machine operating panel before deselecting the handwheel. Proceed as follows:

- ▶ Press the **CTRL** and Handwheel keys on the HR 5xx. The TNC shows the soft-key menu for selecting the potentiometers on the handwheel display
- ▶ Press the **KBD** soft key to activate the potentiometers of the machine operating panel

Incremental jog positioning

With incremental jog positioning the TNC moves the currently active handwheel axis by a preset distance defined by you:

- ▶ Press the handwheel soft key F2 (**STEP**)
- ▶ Activate incremental jog positioning: Press handwheel soft key 3 (**ON**)
- ▶ Select the desired jog increment by pressing the F1 or F2 key. If you press and hold the respective key, each time it reaches a decimal value 0 the TNC increases the counting increment by a factor of 10. If in addition you press the **CTRL** key, the counting increment increases to 1. The smallest possible jog increment is 0.0001 mm. The largest possible is 10 mm
- ▶ Confirm the selected jog increment with soft key 4 (**OK**)
- ▶ With the + or – handwheel key, move the active handwheel axis in the corresponding direction

Entering miscellaneous functions M

- ▶ Press the handwheel soft key F3 (**MSF**)
- ▶ Press the handwheel soft key F1 (**M**)
- ▶ Select the desired M function number by pressing the F1 or F2 key
- ▶ Execute the M function with the NC start key

Entering the spindle speed S

- ▶ Press the handwheel soft key F3 (**MSF**)
- ▶ Press the handwheel soft key F2 (**S**)
- ▶ Select the desired speed by pressing the F1 or F2 key. If you press and hold the respective key, each time it reaches a decimal value 0 the TNC increases the counting increment by a factor of 10. If in addition you press the **CTRL** key, the counting increment increases to 1000
- ▶ Activate the new speed S with the NC start key

Manual operation and setup

15.2 Moving the machine axes

Entering the feed rate F

- ▶ Press the handwheel soft key F3 (**MSF**)
- ▶ Press the handwheel soft key F3 (**F**)
- ▶ Select the desired feed rate by pressing the F1 or F2 key. If you press and hold the respective key, each time it reaches a decimal value 0 the TNC increases the counting increment by a factor of 10. If in addition you press the **CTRL** key, the counting increment increases to 1000
- ▶ Confirm the new feed rate F with the handwheel soft key F3 (**OK**)

Datum setting

- ▶ Press the handwheel soft key F3 (**MSF**)
- ▶ Press the handwheel soft key F4 (**PRS**)
- ▶ If required, select the axis in which the datum is to be set.
- ▶ Reset the axis with the handwheel soft key F3 (**OK**), or with F1 and F2 set the desired value and then confirm with F3 (**OK**)
By also pressing the **CTRL** key, you can increase the counting increment to 10

Changing modes of operation

With the handwheel soft key F4 (**OPM**), you can use the handwheel to switch the mode of operation, provided that the current status of the control allows a mode change.

- ▶ Press the handwheel soft key F4 (**OPM**)
- ▶ Select the desired operating mode by handwheel soft key
 - MAN: Manual Operation
 - MDI: Positioning with manual data input
 - SGL: Program run, single block
 - RUN: Program run, full sequence

Generating a complete traversing block



Your machine tool builder can assign any function to the "Generate NC block" handwheel key. Refer to your machine manual.

- ▶ Select the **Positioning with MDI** operating mode
- ▶ If required, use the arrow keys on the TNC keyboard to select the NC block after which the new traversing block is to be inserted.
- ▶ Activate the handwheel
- ▶ Press the "Generate NC block" handwheel key: The TNC inserts a complete traversing block containing all axis positions selected through the MOD function.

Features in the program run modes of operation

You can use the following functions in the Program Run modes of operation:

- NC start (handwheel NC-start key)
- NC stop (handwheel NC-stop key)
- After the NC-stop key has been pressed: Internal stop (handwheel soft keys **MOP** and then **STOP**)
- After the NC-stop key has been pressed: Manual axis traverse (handwheel soft keys **MOP** and then **MAN**)
- Returning to the contour after the axes were moved manually during a program interruption (handwheel soft keys **MOP** and then **REPO**). Operation is by handwheel soft keys, which function similarly to the control-screen soft keys, see "Returning to the contour", page 607
- On/off switch for the Tilted Working Plane function (handwheel soft keys **MOP** and then **3D**)

Manual operation and setup

15.3 Spindle speed S, feed rate F and miscellaneous function M

15.3 Spindle speed S, feed rate F and miscellaneous function M

Application

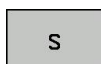
In the **Manual Operation** and **El. Handwheel** operating modes, you can enter the spindle speed S, feed rate F and the miscellaneous functions M with soft keys. The miscellaneous functions are described in page 368.



The machine tool builder determines which miscellaneous functions M are available on your control and what effects they have.

Entering values

Spindle speed S, miscellaneous function M



- ▶ Enter the spindle speed: Press the S soft key

SPINDLE SPEED S=



- ▶ Enter **1000** (spindle speed) and confirm your entry with the machine START button.

The spindle speed S with the entered rpm is started with a miscellaneous function M. Proceed in the same way to enter a miscellaneous function M.

Feed rate F

After entering a feed rate F, confirm your entry with the **ENT** key.

The following is valid for feed rate F:

- If you enter F=0, then the lowest feed rate from the machine parameter **manualFeed** is effective.
- If the feed rate entered exceeds the value defined in the machine parameter **maxFeed**, then the parameter value is effective.
- F is not lost during a power interruption
- The control displays the feed rate.

Spindle speed S, feed rate F and miscellaneous function M 15.3

Adjusting spindle speed and feed rate

With the override knobs you can vary the spindle speed S and feed rate F from 0% to 150% of the set value.



The override knob for spindle speed is only functional on machines with infinitely variable spindle drive.



Activating feed-rate limitation



The feed-rate limit depends on the machine. Refer to your machine manual.

When the F LIMITED soft key is set to ON, the TNC limits the maximum permissible axis speed to the safely limited speed specified by the machine manufacturer.



- Select the **Manual Operation** mode



- Scroll to the last soft-key row



- Switch on/off feed rate limit

Manual operation and setup

15.4 Optional safety concept (Functional safety FS)

15.4 Optional safety concept (Functional safety FS)

Miscellaneous



You machine tool builder adapts the HEIDENHAIN safety concept to your machine. Refer to your machine manual.

Every machine tool operator is exposed to certain risks. Although protective devices can prevent access to dangerous points, the operator must also be able to work on the machine without this protection (e.g. protective door opened). Several guidelines and regulations to minimize these risks have been developed within the last few years.

The HEIDENHAIN safety concept integrated in the TNC controls complies with **Performance Level d** as per EN 13849-1 and SIL 2 as per IEC 61508, features safety-related modes of operation in accordance with EN 12417, and assures extensive operator protection.

The basis of the HEIDENHAIN safety concept is the dual-channel processor structure, which consists of the main computer (MC) and one or more drive controller modules (CC= control computing unit). All monitoring mechanisms are designed redundantly in the control systems. Safety-relevant system data are subject to a mutual cyclic data comparison. Safety-relevant errors always lead to safe stopping of all drives through defined stop reactions.

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

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

Explanation of terms

Safety-related operating modes

Description	Brief description
SOM_1	Safe operating mode 1: automatic operation, production mode
SOM_2	Safe operating mode 2: set-up mode
SOM_3	Safe operating mode 3: manual intervention; only for qualified operators
SOM_4	Safe operating mode 4: Advanced manual intervention, process monitoring

Safety functions

Description	Brief description
SS0, SS1, SS1F, SS2	Safe stop: safe stopping of all drives using different methods
STO	Safe torque off: energy supply to the motor is interrupted. Provides protection against unexpected start of the drives
SOS	Safe operating stop. Provides protection against unexpected start of the drives
SLS	Safely-limited speed. Prevents the drives from exceeding the specified speed limits when the protective door is opened

Manual operation and setup

15.4 Optional safety concept (Functional safety FS)

Checking the axis positions



This function must be adapted to the TNC by your machine manufacturer. Refer to your machine manual.

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

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

- ▶ Select the **Manual Operation** mode
- ▶ Execute the approach with NC Start to move the axes in the sequence shown
- ▶ When the test position has been reached, the TNC asks whether the position was approached correctly: Confirm with the OK soft key if the TNC approached the test position correctly, and with END if the TNC approached the position incorrectly.
- ▶ If you confirmed with OK, you must confirm the correctness of the test position again with the permissive button on the machine operating panel
- ▶ Repeat this procedure for all axes that you want to move to the test position



Danger of collision!

Approach the test positions in such a way that no collision between tool and the workpiece or the clamping devices can occur. If necessary, pre-position the axes manually.



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

Activating feed-rate limitation

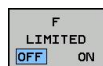
When the F LIMITED soft key is set to ON, the TNC limits the maximum permissible axis speeds to the specified, safely limited speed.



- ▶ Select the **Manual Operation** mode



- ▶ Scroll to the last soft-key row







- ▶ Switch on/off feed rate limit

Additional status displays

On a control with functional safety FS, the general status display contains additional information about the current status of safety functions. The TNC shows this information as operating statuses of the status displays **T**, **S** and **F**

Status display	Brief description
STO	Energy supply to the spindle or a feed drive is interrupted.
SLS	Safely-limited speed: A safely limited speed is active.
SOS	Safe operating stop: Safe operating stop is active.
STO	Safe torque off: Energy supply to the motor is interrupted.

The TNC shows the active safety-related mode of operation with an icon in the header to the right of the operating mode text:

Button	Safety-related operating mode
	SOM_1 operating mode active
	SOM_2 mode active
	SOM_3 mode active
	SOM_4 mode active

Manual operation and setup

15.5 Datum management with the preset table

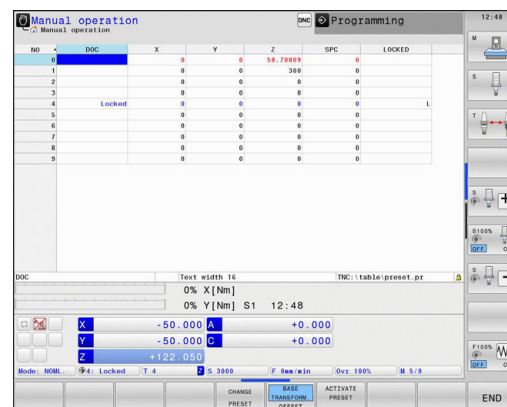
15.5 Datum management with the preset table

Note



You should definitely use the preset table if:

- Your machine is equipped with rotary axes (tilting table or swivel head) and you work with the function for tilting the working plane
- Your machine is equipped with a spindle-head changing system
- Up to now you have been working with older TNC controls with REF-based datum tables
- You wish to machine identical workpieces that are differently aligned



The preset table can contain any number of lines (datums). To optimize the file size and the processing speed, you should use only as many lines as you need for preset management.

For safety reasons, new lines can be inserted only at the end of the preset table.

Saving the datums in the preset table

The preset table has the name **PRESET.PR**, and is saved in the directory **TNC:\table**. **PRESET.PR** is editable in the **Manual Operation** and **El. Handwheel** modes only if the **CHANGE PRESET** soft key was pressed. You can open the **PRESET.PR** preset table in the **Programming** mode of operation, but you cannot edit it.

It is permitted to copy the preset table into another directory (for data backup). Lines are also always write-protected in the copied tables. You therefore cannot edit them.

Never change the number of lines in the copied tables! That could cause problems when you want to reactivate the table.

To activate the preset table copied to another directory you have to copy it back to the directory **TNC:\table**.

There are several methods for saving datums and/or basic rotations in the preset table:

- Via touch probe cycles in the **Manual Operation** and **El. Handwheel** modes
- Through the probing cycles 400 to 402 and 410 to 419 in automatic mode (see User's Manual, Cycles, Chapters 14 and 15)
- Manual entry (see description below)



Basic rotations from the preset table rotate the coordinate system about the preset, which is shown in the same line as the basic rotation.

Remember to ensure that the position of the tilting axes matches the corresponding values of the 3-D ROT menu when setting the datum. Therefore:

- If the "Tilt working plane" function is not active, the position display for the rotary axes must be = 0° (zero the rotary axes if necessary).
- If the "Tilt working plane" function is active, the position displays for the rotary axes must match the angles entered in the 3-D ROT menu.

PLANE RESET does **not** reset the active 3D-ROT.

The line 0 in the preset table is write protected. In line 0, the TNC always saves the datum that you most recently set manually via the axis keys or via soft key. If the datum set manually is active, the TNC displays the text **PR MAN(0)** in the status display.

Manual operation and setup

15.5 Datum management with the preset table

Manually saving the datums in the preset table

In order to save datums in the preset table, proceed as follows:



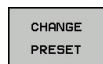
- Select the **Manual Operation** mode



- Move the tool slowly until it touches (scratches) the workpiece surface, or position the measuring dial correspondingly



- Display the preset table: The TNC opens the preset table and sets the cursor to the active table row



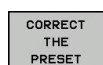
- Select functions for entering the presets: The TNC displays the available possibilities for entry in the soft-key row. See the table below for a description of the entry possibilities



- Select the line in the preset table that you want to change (the line number is the preset number)



- If needed, select the column (axis) in the preset table that you want to change



- Use the soft keys to select one of the available entry possibilities (see the following table)

Soft key

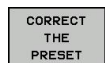
Function



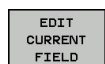
Directly transfer the actual position of the tool (the measuring dial) as the new preset: This function only saves the preset in the axis which is currently highlighted





Assign any value to the actual position of the tool (the measuring dial): This function only saves the preset in the axis which is currently highlighted. Enter the desired value in the pop-up window



Incrementally shift a preset already stored in the table: This function only saves the preset in the axis which is currently highlighted. Enter the desired corrective value with the correct sign in the pop-up window. If inch display is active: Enter the value in inches, and the TNC will internally convert the entered values to mm








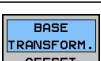
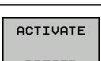
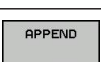
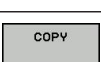

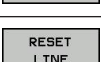
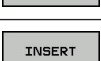

Directly enter the new datum without calculation of the kinematics (axis-specific). Only use this function if your machine has a rotary table, and you want to set the datum to the center of the rotary table by entering 0. This function only saves the datum in the axis which is currently highlighted. Enter the desired value in the pop-up window. If inch display is active: Enter the value in inches, and the TNC will internally convert the entered values to mm

Soft key	Function
	Select the BASIC TRANSFORMATION/AXIS OFFSET view. The BASIC TRANSFORMATION view shows the X, Y and Z columns. Depending on the machine, the SPA, SPB and SPC columns are displayed additionally. Here, the TNC saves the basic rotation (for the Z tool axis, the TNC uses the SPC column). The OFFSET view shows the offset values for the preset
	Write the currently active datum to a selectable line in the table: This function saves the datum in all axes, and then activates the appropriate row in the table automatically. If inch display is active: Enter the value in inches, and the TNC will internally convert the entered values to mm

Manual operation and setup

15.5 Datum management with the preset table

Editing the preset table

Soft key	Editing function in table mode
	Select the table start
	Select the table end
	Select the previous page in the table
	Select the next page in the table
	Select the functions for preset entry
	Display the "Basic Transformation/Axis Offset" selection
	Activate the datum of the selected line of the preset table
	Add the entered number of lines to the end of the table (2nd soft-key row)
	Copy the highlighted field (2nd soft-key row)
	Insert the copied field (2nd soft-key row)
	Reset the selected line: The TNC enters - in all columns (2nd soft-key row)
	Insert a single line at the end of the table (2nd soft-key row)
	Delete a single line at the end of the table (2nd soft-key row)

Overwrite protection for datum

The line 0 in the preset table is write protected. The TNC saves the last manually set datum in line 0.

You can overwrite-protect further lines in the preset table with the **LOCKED** column. The overwrite-protected lines are color-highlighted in the preset table.



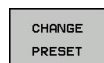
Caution: Data may be lost!

If you forget the password, you can no longer reset the write-protection of a password-protected line.

Make note of the password when you password-protect lines.

Preferentially use simple protection with the soft key.

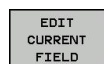
Proceed as follows to protect a datum from overwriting:



- ▶ Press the **CHANGE PRESET** soft key

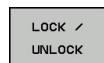


- ▶ Select the **LOCKED** column



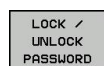
- ▶ Press the **EDIT CURRENT FIELD** soft key

Protection for datum without using password:

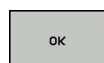


- ▶ Press the soft key. The TNC writes an **L** in the LOCKED column.

Protection for datum with password:



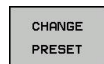
- ▶ Press the soft key



- ▶ Enter the password into the pop-up window
- ▶ Confirm with the **OK** soft key or the **ENT** key: The TNC writes **###** in the LOCKED column.

Rescind write-protection

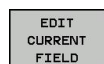
To edit a line you have previously write-protected, proceed as follows:



- ▶ Press the **CHANGE PRESET** soft key



- ▶ Select the **LOCKED** column

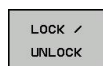


- ▶ Press the **EDIT CURRENT FIELD** soft key

Manual operation and setup

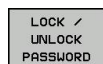
15.5 Datum management with the preset table

Datum protected without password:

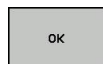


- ▶ Press the **LOCK / UNLOCK** soft key. The TNC rescinds write-protection.

Datum protected with password:



- ▶ Press the **LOCK / UNLOCK PASSWORD** soft key



- ▶ Enter the password into the pop-up window
- ▶ Confirm with the **OK** soft key or the **ENT** key: The TNC rescinds write-protection.

Activating the datum

Activating a datum from the preset table in the Manual Operation mode



When activating a datum from the preset table, the TNC resets the active datum shift, mirroring, rotation and scaling factor.

However, a coordinate transformation that was programmed in Cycle 19, Tilted Working Plane, or through the PLANE function, remains active.



- ▶ Select the **Manual Operation** mode



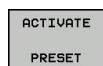
- ▶ Display the preset table



- ▶ Select the datum number you want to activate, or



- ▶ With the **GOTO** key, select the datum number that you want to activate. Confirm with the **ENT** key



- ▶ Activate the datum



- ▶ Confirm activation of the datum. The TNC sets the display and—if defined—the basic rotation



- ▶ Exit the preset table

Activating a datum from the preset table in an NC program

Use Cycle 247 in order to activate datums from the preset table during program run. In Cycle 247 you simply define the number of the datum to be activated (see User's Manual for Cycles, Cycle 247 DATUM SETTING).

15.6 Datum setting without a 3-D touch probe

Note



Setting the datum with a 3-D touch probe: see "Datum setting with 3-D touch probe", page 563.

You fix a datum by setting the TNC position display to the coordinates of a known position on the workpiece.

Preparation

- ▶ Clamp and align the workpiece
- ▶ Insert the zero tool with known radius into the spindle
- ▶ Ensure that the TNC is showing the actual position values

Setting datum with an end mill



Protective measure

If the workpiece surface must not be scratched, you can lay a metal shim of known thickness d on it. Then enter a tool axis datum value that is larger than the desired datum by the value d .



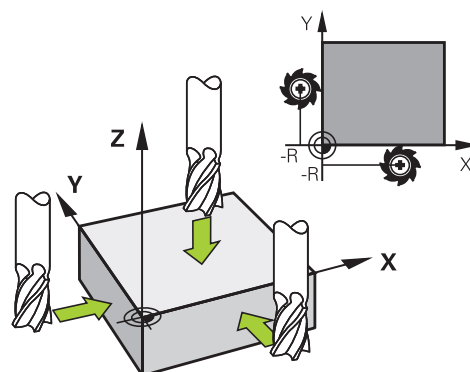
- ▶ Select the **Manual Operation** mode



- ▶ Move the tool slowly until it touches (scratches) the workpiece surface



- ▶ Select the axis



DATUM SETTING Z=



- ▶ Zero tool in spindle axis: Set the display to a known workpiece position (e.g. 0) or enter the thickness d of the shim. In the tool axis, offset the tool radius



Repeat the process for the remaining axes.

If you are using a preset tool, set the display of the tool axis to the length L of the tool or enter the sum $Z=L+d$



The TNC automatically saves the datum set with the axis keys in line 0 of the preset table.

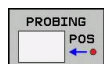
Manual operation and setup

15.6 Datum setting without a 3-D touch probe

Using touch probe functions with mechanical probes or measuring dials

If you do not have an electronic 3-D touch probe on your machine, you can also use all the previously described manual touch probe functions (exception: calibration function) with mechanical probes or by simply touching the workpiece with the tool, see page 545.

In place of the electronic signal generated automatically by a 3-D touch probe during probing, you can manually initiate the trigger signal for capturing the **probing position** by pressing a key. Proceed as follows:



- ▶ Select any touch probe function by soft key
- ▶ Move the mechanical probe to the first position to be captured by the TNC
- ▶ Confirm the position: Press the actual-position-capture soft key for the TNC to save the current position
- ▶ Move the mechanical probe to the next position to be captured by the TNC
- ▶ Confirm the position: Press the actual-position-capture soft key for the TNC to save the current position
- ▶ If required, move to additional positions and capture as described previously
- ▶ **Datum:** In the menu window, enter the coordinates of the new datum, confirm with the **SET DATUM** soft key, or write the values to a table (see "Writing measured values from the touch probe cycles in a datum table", page 551, or see "Writing measured values from the touch probe cycles in the preset table", page 552)
- ▶ Terminate the probing function: Press the **END** key

15.7 Using 3-D touch probes

Overview


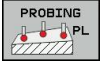
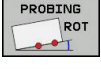
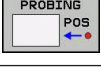
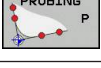
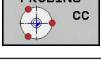
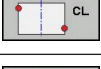

The following touch probe cycles are available in the **Manual Operation** mode:



HEIDENHAIN only gives warranty for the function of the probing cycles if HEIDENHAIN touch probes are used.



The TNC must be specially prepared by the machine tool builder for the use of a 3-D touch probe. Refer to your machine manual.

Soft key	Function	Page
	Calibrating the 3-D Touch Probe	553
	Measuring a 3-D basic rotation by probing a plane	561
	Measuring a basic rotation using a line	560
	Setting a datum in any axis	563
	Setting a corner as datum	564
	Setting a circle center as datum	565
	Setting the centerline as datum	567
	Touch probe system data management	See User's Manual for Cycles

15.7 Using 3-D touch probes

You can also use all manual touch probe cycles, except the corner probing cycle and plane probing cycle, in Turning mode. Observe that in Turning mode all measured data in the X coordinate are calculated and displayed as diameter values.

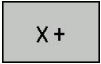

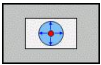
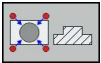
To use the touch probe in Turning mode you should separately calibrate the touch probe in Turning mode. Because the basic setting of the turning spindle in Milling mode and Turning mode may deviate, you should calibrate the touch probe without center offset. You can create additional tool data for this purpose, e.g. as an indexed tool.



For more information about the touch probe table, refer to the User's Manual for Cycle Programming.

Functions in touch probe cycles

Soft keys that are used to select the probing direction or a probing routine are displayed in the manual touch probe cycles. The soft keys displayed vary depending on the respective cycle:

Soft key	Function
	Select the probing direction
	Capture the actual position
	Probe hole (inside circle) automatically
	Probe stud (outside circle) automatically

Automatic probing routine for holes and studs



If you use a function for probing a circle automatically, the TNC automatically positions the touch probe to the respective touch points. Ensure that the positions can be approached without collision.

If you use a probing routine for probing a hole or a stud automatically, the TNC opens a form with the required input fields.

Input fields in the Measure stud and Measure hole forms

Input field	Function
Stud diameter? or Hole diameter?	Diameter of probe contact (optional for holes)
Safety clearance?	Distance to the probe contact in the plane
Incr. clearance height?	Positioning of touch probe in spindle axis direction (starting from the current position)
Starting angle?	Angle for the first probing operation (0° = Positive direction of principal axis, i.e. in X+ for spindle axis Z). All other probe angles result from the number of touch points.
Number of touch points?	Number of probing operations (3 to 8)
Angular length?	Probing a full circle (360°) or a circle segment (angular length < 360°)

Manual operation and setup

15.7 Using 3-D touch probes

Position the touch probe approximately in the center of the hole (inside circle) or near the first touch point on the stud (outside circle), and select the soft key for the first probing direction. Once you press the machine START button to start the touch probe cycle, the TNC automatically performs all prepositioning movements and probing operations.

The TNC positions the touch probe to the individual touch points, taking the safety clearance into account. If a clearance height has been defined, the TNC positions the touch probe to clearance height in the spindle axis beforehand.

The TNC approaches the position at the feed rate **FMAX** defined in the touch probe table. The defined probing feed rate **F** is used for the actual probing operation.



Before starting the automatic probing routine, you need to preposition the touch probe near the first touch point. Offset the touch probe by approximately the safety clearance (value from touch probe table + value from input form) opposite to the probing direction.

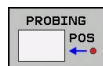
For an inside circle with a large diameter, the TNC can also preposition the touch probe on a circular arc at the positioning feed rate FMAX. This requires that you enter a safety clearance for prepositioning and the hole diameter in the input form. Position the touch probe inside the hole at a position that is offset by approximately the safety clearance from the wall. For prepositioning, keep in mind the starting angle for the first probing operation (with an angle of 0°, the TNC probes in the positive direction of the principal axis).

Selecting touch probe cycles

- ▶ Select the **Manual Operation** or **El. Handwheel** mode of operation



- ▶ Select the touch probe functions by pressing the **TOUCH PROBE** soft key. The TNC displays additional soft keys (see overview table).



- ▶ Select the touch probe cycle by pressing the appropriate soft key, for example **PROBING POS**, for the TNC to display the associated menu



When you select a manual probing function, the TNC opens a form displaying all data required. The content of the forms varies depending on the respective function.

You can also enter values in some of the fields. Use the arrow keys to move to the desired input field. You can position the cursor only in fields that can be edited. Fields that cannot be edited appear dimmed.

Manual operation and setup

15.7 Using 3-D touch probes

Recording measured values from the touch-probe cycles



The TNC must be specially prepared by the machine tool builder for use of this function. Refer to your machine manual.

After executing any selected touch probe cycle, the TNC displays the soft key **WRITE LOG TO FILE**. If you press this soft key, the TNC will record the current values determined in the active touch probe cycle.

If you store the measuring results, the TNC creates the text file TCHPRMAN.TXT. If you have not defined a path in the machine parameter **fn16DefaultPath**, the TNC will store the TCHPRMAN.TXT and TCHPRMAN.html files in the main directory **TNC:**.



When you press the **WRITE LOG TO FILE** soft key, the TCHPRMAN.TXT file must not be active in the **Programming** mode of operation. The TNC will otherwise display an error message.

The TNC writes the measured values to the TCHPRMAN.TXT or TCHPRMAN.html file. If you execute several touch probe cycles in succession and want to store the resulting measured data, you must make a backup of the contents stored in TCHPRMAN.TXT between the individual cycles by copying or renaming the file.

Format and content of the TCHPRMAN.TXT file are preset by the machine tool builder.

Writing measured values from the touch probe cycles in a datum table



Use this function if you want to save measured values in the workpiece coordinate system. If you want to save measured values in the machine-based coordinate system (REF coordinates), press the **ENTER IN PRESET TABLE SOFT KEY**, see "Writing measured values from the touch probe cycles in the preset table", page 552.

With the **ENTER IN DATUM TABLE** soft key, the TNC can write the values measured during a touch probe cycle in a datum table:

- ▶ Select any probe function
- ▶ Enter the desired coordinates of the datum in the appropriate input boxes (depends on the touch probe cycle being run)
- ▶ Enter the datum number in the **Number in table=** input box
- ▶ Press the **ENTER IN DATUM TABLE** soft key. The TNC saves the datum in the indicated datum table under the entered number

15.7 Using 3-D touch probes

Writing measured values from the touch probe cycles in the preset table



Use this function if you want to save measured values in the machine-based coordinate system (REF coordinates). If you want to save measured values in the workpiece coordinate system, use the **ENTER IN DATUM TABLE SOFT KEY**, see "Writing measured values from the touch probe cycles in a datum table", page 551.

With the **ENTER IN PRESET TABLE** soft key, the TNC can write the values measured during a probe cycle in the preset table. The measured values are then stored referenced to the machine-based coordinate system (REF coordinates). The preset table has the name PRESET.PR, and is saved in the directory TNC:\table\.

- ▶ Select any probe function
- ▶ Enter the desired coordinates of the datum in the appropriate input boxes (depends on the touch probe cycle being run)
- ▶ Enter the preset number in the **Number in table:** input box
- ▶ Press the **ENTER IN PRESET TABLE** soft key. The TNC saves the datum in the preset table under the entered number

15.8 Calibrating a 3-D touch trigger probe

Introduction

In order to precisely specify the actual trigger point of a 3-D touch probe, you must calibrate the touch probe, otherwise the TNC cannot provide precise measuring results.



Always calibrate a touch probe in the following cases:

- Commissioning
- Stylus breakage
- Stylus exchange
- Change in the probe feed rate
- Irregularities caused, for example, when the machine heats up
- Change of active tool axis

When you press the **OK** soft key after calibration, the calibration values are applied to the active touch probe. The updated tool data become effective immediately, and a new tool call is not necessary.

During calibration, the TNC finds the effective length of the stylus and the effective radius of the ball tip. To calibrate the 3-D touch probe, clamp a ring gauge or a stud of known height and known radius to the machine table.





The TNC provides calibration cycles for calibrating the length and the radius:

- Press the **TOUCH PROBE** soft key



- Display the calibration cycles: Press **CALIBRATE TS**.
- Select the calibration cycle

Calibration cycles of the TNC

Soft key	Function	Page
	Calibrating the length	554
	Measure the radius and the center offset using a calibration ring	page 556
	Measure the radius and the center offset using a stud or a calibration pin	page 556
	Measure the radius and the center offset using a calibration sphere	page 557

Manual operation and setup

15.8 Calibrating a 3-D touch trigger probe

Calibrating the effective length

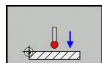


HEIDENHAIN only gives warranty for the function of the probing cycles if HEIDENHAIN touch probes are used.

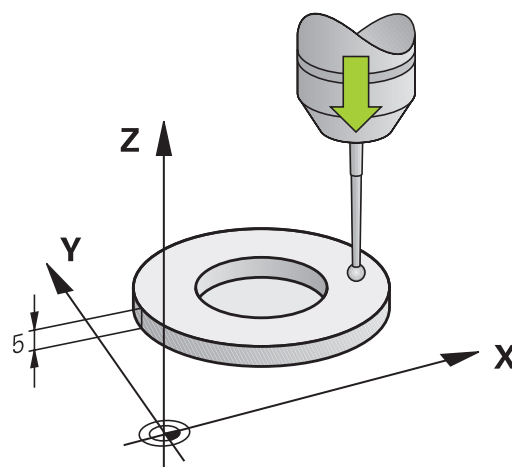


The effective length of the touch probe is always referenced to the tool datum. The machine tool builder usually defines the spindle tip as the tool datum.

- Set the datum in the spindle axis such that for the machine tool table $Z=0$.



- Select the calibration function for the touch probe length: Press the **CAL. L** soft key. The TNC displays the current calibration data.
- Datum for length: Enter the height of the ring gauge in the menu window
- Move the touch probe to a position just above the ring gauge
- To change the traverse direction (if necessary), press a soft key or an arrow key
- To probe the upper surface of the ring gauge, press the machine START button
- Checking the results
- Press the **OK** soft key for the values to take effect
- Press the **CANCEL** soft key to terminate the calibrating function. The TNC logs the calibration process in TCHPRMAN.html.



Calibrating the effective radius and compensating center misalignment

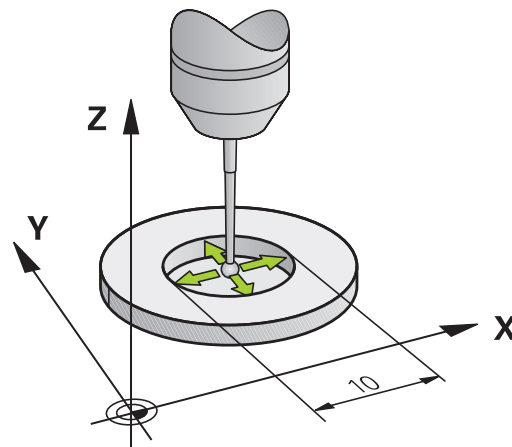


HEIDENHAIN only gives warranty for the function of the probing cycles if HEIDENHAIN touch probes are used.



The center offset can be determined only with a suitable touch probe.

If you want to calibrate using the outside of an object, you need to preposition the touch probe above the center of the calibration sphere or calibration pin. Ensure that the touch points can be approached without collision.



When calibrating the ball tip radius, the TNC executes an automatic probing routine. During the first probing cycle, the TNC determines the center of the calibration ring or stud (coarse measurement) and positions the touch probe in the center. Then the ball tip radius is determined during the actual calibration process (fine measurement). If the touch probe allows probing from opposite orientations, the center offset is determined during another cycle.

The characteristic of whether and how your touch probe can be oriented is already defined in HEIDENHAIN touch probes. Other touch probes are configured by the machine tool builder.

After the touch probe is inserted, it normally needs to be aligned exactly with the spindle axis. The calibration function can determine the offset between touch-probe axis and spindle axis by probing from opposite orientations (rotation by 180°) and can compute the compensation.

The calibration routine varies depending on how your touch probe can be oriented:

- No orientation possible or orientation possible in only one direction: The TNC executes one approximate and one fine measurement and determines the effective ball tip radius (column R in tool.t)
- Orientation possible in two directions (e.g., HEIDENHAIN touch probes with cable): The TNC executes one approximate and one fine measurement, rotates the touch probe by 180° and then executes one more probing operation. The center offset (CAL_OF in tchprobe.tp) is determined in addition to the radius by probing from opposite orientations.
- Any orientation possible (e.g. HEIDENHAIN infrared touch probes): For probing routine, see "orientation possible in two directions."

Manual operation and setup

15.8 Calibrating a 3-D touch trigger probe

Calibration using a calibration ring

Proceed as follows for manual calibration using a calibration ring:

- ▶ In the **Manual Operation** mode, position the ball tip inside the bore of the ring gauge



- ▶ Select the calibration function: Press the **CAL. R** soft key. The TNC displays the current calibration data.
- ▶ Enter the diameter of the ring gauge
- ▶ Enter the start angle
- ▶ Enter the number of touch points
- ▶ Start the probing procedure: Press the machine START button. The 3-D touch probe probes all required touch points in an automatic probing routine and calculates the effective ball-tip radius. If probing from opposite orientations is possible, the TNC calculates the center offset
- ▶ Checking the results
- ▶ Press the **OK** soft key for the values to take effect
- ▶ Press the **END** soft key to terminate the calibrating function. The TNC logs the calibration process in TCHPRMAN.html.

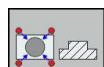


In order to be able to determine the ball-tip center misalignment, the TNC needs to be specially prepared by the machine manufacturer. Refer to your machine manual.

Calibration with a stud or calibration pin

Proceed as follows for manual calibration with a stud or calibration pin:

- ▶ In the **Manual operation** mode, position the ball tip above the center of the calibration pin



- ▶ Select the calibration function: Press the **CAL. R** soft key
- ▶ Enter the diameter of the stud
- ▶ Enter the safety clearance
- ▶ Enter the start angle
- ▶ Enter the number of touch points
- ▶ Start the probing procedure: Press the machine START button. The 3-D touch probe probes all required touch points in an automatic probing routine and calculates the effective ball-tip radius. If probing from opposite orientations is possible, the TNC calculates the center offset
- ▶ Checking the results
- ▶ Press the **OK** soft key for the values to take effect
- ▶ Press the **END** soft key to terminate the calibrating function. The TNC logs the calibration process in TCHPRMAN.html.

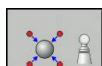


In order to be able to determine the ball-tip center misalignment, the TNC needs to be specially prepared by the machine manufacturer.
Refer to your machine manual.

Calibration using a calibration sphere

Proceed as follows for manual calibration using a calibration sphere:

- ▶ In the **Manual Operation** mode, position the ball tip above the center of the calibration sphere



- ▶ Select the calibration function: Press the **CAL. R** soft key
- ▶ Enter the diameter of the sphere
- ▶ Enter the safety clearance
- ▶ Enter the start angle
- ▶ Enter the number of touch points
- ▶ Select Length measurement, if applicable
- ▶ Enter Datum for length, if applicable
- ▶ Start the probing procedure: Press the machine START button. The 3-D touch probe probes all required touch points in an automatic probing routine and calculates the effective ball-tip radius. If probing from opposite orientations is possible, the TNC calculates the center offset
- ▶ Checking the results
- ▶ Press the **OK** soft key for the values to take effect
- ▶ Press the **END** soft key to terminate the calibrating function. The TNC logs the calibration process in TCHPRMAN.html.



In order to be able to determine the ball-tip center misalignment, the TNC needs to be specially prepared by the machine manufacturer.
Refer to your machine manual.


Manual operation and setup

15.8 Calibrating a 3-D touch trigger probe


Displaying calibration values

The TNC saves the effective length and effective radius of the touch probe in the tool table. The TNC saves the ball-tip center misalignment in the touch-probe table, in the **CAL_OF1** (principal axis) and **CAL_OF2** (minor axis) columns. You can display the values on the screen by pressing the **TOUCH PROBE TABLE** soft key.

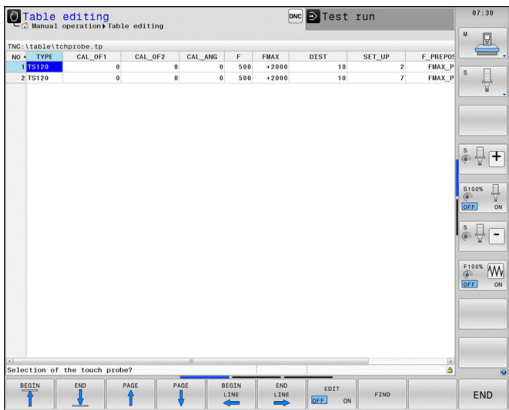
During calibration, the TNC automatically creates the TCHPRMAN.html log file to which the calibration values are saved.



Make sure that you have activated the correct tool number before using the touch probe, regardless of whether you wish to run the touch probe cycle in automatic mode or in the **Manual Operation** operating mode.



For more information about the touch probe table, refer to the User’s Manual for Cycle Programming.



15.9 Compensating workpiece misalignment with 3-D touch probe

Introduction



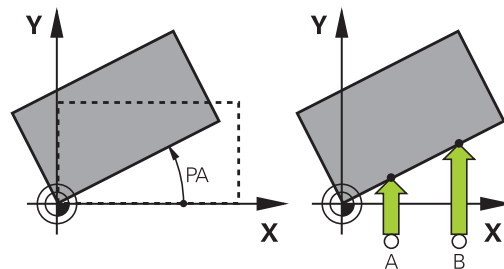
HEIDENHAIN only gives warranty for the function of the probing cycles if HEIDENHAIN touch probes are used.

The TNC electronically compensates workpiece misalignment by computing a "basic rotation."

For this purpose, the TNC sets the rotation angle to the desired angle with respect to the reference axis in the working plane. See figure at right.

The TNC interprets the measured angle as rotation around the tool direction in the workpiece coordinate system, and saves the values in the columns SPA, SPB and SPC of the preset table.

To identify the basic rotation, probe two points on the side of the workpiece. The sequence in which you probe the points influences the calculated angle. The measured angle goes from the first to the second probing point. You can also identify the basic rotation by holes or studs.



Select the probe direction perpendicular to the angle reference axis when measuring workpiece misalignment.

To ensure that the basic rotation is calculated correctly during program run, program both coordinates of the working plane in the first positioning block.

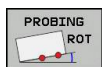
You can also use a basic rotation in conjunction with the PLANE function. In this case, first activate the basic rotation and then the PLANE function.

You can also activate a basic rotation without probing a workpiece. For this purpose enter a value in the basic rotation menu and press the **SET BASIC ROTATION** soft key.

Manual operation and setup

15.9 Compensating workpiece misalignment with 3-D touch probe

Identifying basic rotation



- ▶ Select the probe function by pressing the **PROBING ROT** soft key
- ▶ Position the touch probe at a position near the first touch point
- ▶ Select the probe direction or probing routine by soft key
- ▶ Start the probing procedure: Press the machine START button
- ▶ Position the touch probe at a position near the second touch point
- ▶ To probe the workpiece, press the machine START button. The TNC determines the basic rotation and displays the angle after the dialog **Rotation angle**
- ▶ Activate basic rotation: Press the **SET BASIC ROTATION** soft key
- ▶ Terminate the probe function by pressing the END soft key.

The TNC logs the probing process in TCHPRMAN.html.

Saving a basic rotation in the preset table

- ▶ After the probing process, enter the preset number in which the TNC is to save the active basic rotation in the **Number in table:** input box
- ▶ Press the **BASIC ROT. IN PRESETTAB.** soft key to save the basic rotation in the preset table

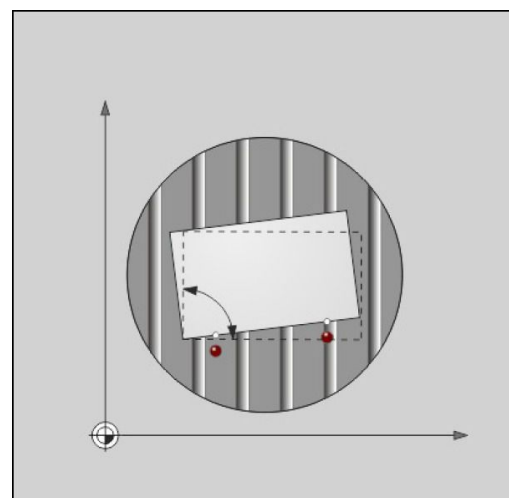
Compensation of workpiece misalignment by rotating the table

- ▶ To compensate the identified misalignment by a rotary table position, press the **ALIGN ROTARY TABLE** soft key after the probing process



Position all axes to avoid a collision before table rotation. The TNC outputs an additional warning before table rotation.

- ▶ If you want to set the datum in the rotary table axis, press the **SET TABLE ROTATION** soft key.
- ▶ You can also save the misalignment of the rotary table in any line of the Preset table. Enter the line number and press the **TABLEROT IN PRESETTAB.** soft key. The TNC saves the angle in the offset column of the rotary table, e.g. in the C_OFFS column with a C axis. If necessary, the view in the Preset table has to be changed with the **BASIS-TRANSFORM./OFFSET** soft key to display this column.

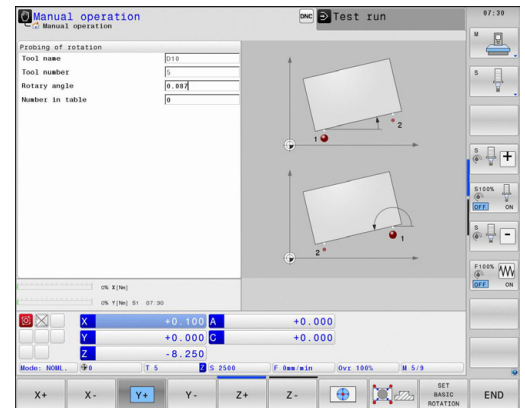


Compensating workpiece misalignment with 3-D touch probe 15.9

Displaying a basic rotation

When you select the **PROBING ROT** function, the TNC displays the active angle of basic rotation in the dialog **Rotation angle**. The TNC also displays the rotation angle in the additional status display (**STATUS POS.**).

In the status display a symbol is shown for a basic rotation whenever the TNC is moving the machine axes according to a basic rotation.



Canceling a basic rotation

- ▶ Select the probe function by pressing the **PROBING ROT** soft key
- ▶ Enter a rotation angle of zero and confirm with the **SET BASIC ROTATION** soft key
- ▶ Terminate the probe function by pressing the **END** soft key

Measuring 3-D basic rotation

The misalignment of any tilted plane can be measured by probing 3 positions. The **Probe plane** function enables you to measure this misalignment and save it as a 3-D basic rotation in the preset table.



Observe the following when selecting the touch points:

The sequence and position of the touch points determines how the TNC calculates the direction of the plane.

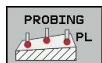
With the first two points you specify the direction of the reference axis. Define the second point in the positive direction of the desired reference axis. The position of the third point determines the direction of the minor axis and tool axis. Define the third point in the positive Y axis of the desired workplace coordinate system.

- 1st point: On the reference axis
- 2nd point: On the reference axis, in a positive direction from the first point
- 3rd point: On the minor axis, in a positive direction of the desired workplace coordinate system

Optionally inputting a datum angle enables you to define the nominal direction of the probed plane.

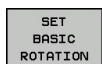
Manual operation and setup

15.9 Compensating workpiece misalignment with 3-D touch probe



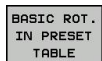
- ▶ Select the probing function: Press the **PROBING PL** soft key. The TNC then displays the current 3-D basic rotation
- ▶ Position the touch probe at a position near the first touch point
- ▶ Select the probe direction or probing routine by soft key
- ▶ Start the probing procedure: Press the machine START button
- ▶ Position the touch probe at a position near the second touch point
- ▶ To probe the workpiece, press the machine START button
- ▶ Position the touch probe near the third touch point
- ▶ Probing: Press the machine START button. The TNC measures the 3-D basic rotation and displays the values for SPA, SPB and SPC related to the active workpiece coordinate system
- ▶ If required, enter the datum angle

Activate 3-D basic rotation



- ▶ Press the **SET BASIC ROTATION** soft key

Saving a 3-D basic rotation in the preset table



- ▶ Press the **BASIC ROT. IN PRESET TABLE** soft key



- ▶ Terminate the probe function by pressing the **END** soft key


The TNC saves the 3-D basic rotation in the columns SPA, SPB or SPC of the preset table.

Aligning 3-D basic rotation

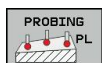
If the machine has two rotary axes and the probed 3-D basic rotation is activated, you can align the rotary axes with reference to the 3-D basic rotation using the **ALIGN ROTARY AXES** soft key. In such cases, Tilted Working Plane becomes active for all machine operating modes.

After aligning the plane, you can align the reference axis with the **Probing rot** function.

Displaying 3-D basic rotation

In the status display the TNC shows the symbol  for the 3-D basic rotation, if a 3-D basic rotation is saved in the active datum. The TNC traverses the machine axes according to the 3-D basic rotation.

Canceling a 3-D basic rotation

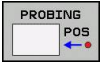

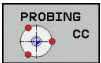
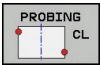


- ▶ Select the probe function by pressing the **PROBING PL** soft key
- ▶ Enter 0 for all angles
- ▶ Press the **SET BASIC ROTATION** soft key
- ▶ Terminate the probe function by pressing the **END** soft key

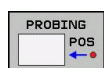
15.10 Datum setting with 3-D touch probe

Overview

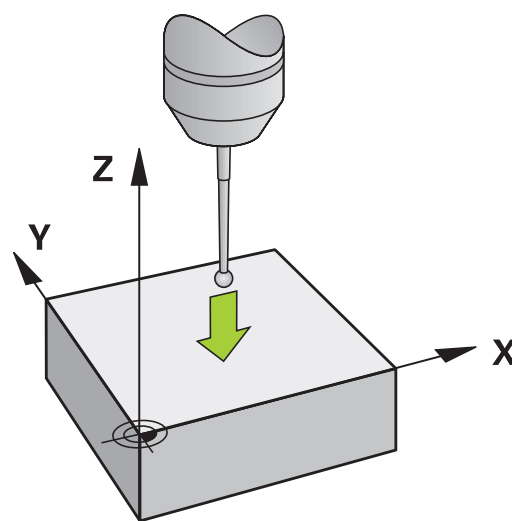
The following soft-key functions are available for setting the datum on an aligned workpiece:

Soft key	Function	Page
	Datum setting in any axis with	563
	Setting a corner as datum	564
	Setting a circle center as datum	565
	Center line as datum Setting the center line as datum	567

Datum setting in any axis



- ▶ Select the probing function: Press the **PROBING POS** soft key
- ▶ Move the touch probe to a position near the touch point
- ▶ Use the soft keys to select the probe axis and direction in which you want to set the preset, such as Z in direction Z–
- ▶ Start the probing procedure: Press the machine START button
- ▶ **Datum:** Enter the nominal coordinate and confirm your entry with the **SET DATUM** soft key, see "Writing measured values from the touch probe cycles in a datum table", page 551
- ▶ To terminate the probe function, press the **END** soft key

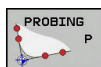


HEIDENHAIN only gives warranty for the function of the probing cycles if HEIDENHAIN touch probes are used.

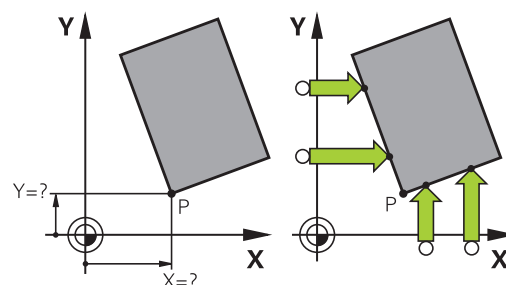
Manual operation and setup

15.10 Datum setting with 3-D touch probe

Corner as datum



- ▶ Select the touch probe function: Press the **PROBING P** soft key
- ▶ Position the touch probe near the first touch point on the first workpiece edge
- ▶ Select the probe direction by soft key
- ▶ Start the probing procedure: Press the machine START button
- ▶ Position the touch probe near the second touch point on the same workpiece edge
- ▶ Start the probing procedure: Press the machine START button
- ▶ Position the touch probe near the first touch point on the second workpiece edge
- ▶ Select the probe direction by soft key
- ▶ Start the probing procedure: Press the machine START button
- ▶ Position the touch probe near the second touch point on the same workpiece edge
- ▶ Start the probing procedure: Press the machine START button
- ▶ **Datum:** Enter both datum coordinates into the menu window, and confirm your entry with the **SET DATUM** soft key, or see "Writing measured values from the touch probe cycles in the preset table", page 552)
- ▶ To terminate the probe function, press the END soft key



HEIDENHAIN only gives warranty for the function of the probing cycles if HEIDENHAIN touch probes are used.



You can identify the intersection of two straight lines by holes or studs and set this as the datum. For each straight line however, probing must only be with two identical touch probe functions (e.g. two holes).

The "Corner as datum" probing cycle identifies the angle and intersection of two straight lines. In addition to datum setting, the cycle can also activate a basic rotation. The TNC has two soft keys for you to decide which straight line you wish to use for this. The soft key **ROT 1** activates the angle of the first straight line as basic rotation and the soft key **ROT 2** the angle of the second straight line.

If you wish to activate the basic rotation in the cycle, you must always do this before datum setting. After you set a datum or write to a zero point or preset table the **ROT 1** and **ROT 2** soft keys are no longer displayed.

Circle center as datum

With this function, you can set the datum at the center of bore holes, circular pockets, cylinders, studs, circular islands, etc.

Inside circle:

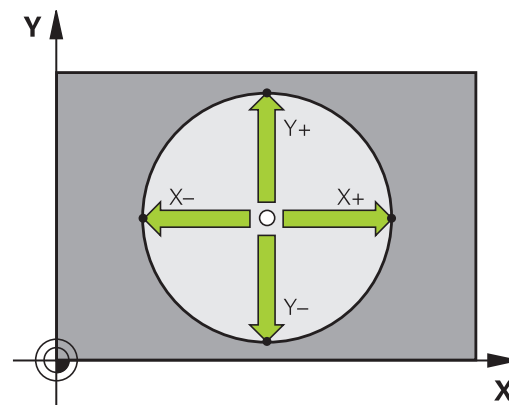
The TNC probes the inside wall of a circle in all four coordinate axis directions.

For incomplete circles (circular arcs) you can choose the appropriate probing direction.

- Position the touch probe approximately in the center of the circle



- Select the touch probe function: Press the **PROBING CC** soft key
- Select the probing direction or press the soft key for the automatic probing routine
- Probing: Press the machine START button. The touch probe probes the inside wall of the circle in the selected direction. If you are not using the automatic probing routine, you need to repeat this procedure. After the third probing operation, you can have the TNC calculate the center (four touch points are recommended)
- Terminate the probing procedure and switch to the evaluation menu: Press the **EVALUATE** soft key
- **Datum:** In the menu window, enter both coordinates of the circle center, confirm with the **SET DATUM** soft key, or write the values to a table (see "Writing measured values from the touch probe cycles in a datum table", page 551, or see "Writing measured values from the touch probe cycles in the preset table", page 552)
- Terminate the probing function: Press the **END** soft key



The TNC needs only three touch points to calculate outside or inside circles, e.g. for circle segments. More precise results are obtained if you measure circles using four touch points, however. You should always preposition the touch probe in the center, or as close to the center as possible.

Manual operation and setup

15.10 Datum setting with 3-D touch probe

Outside circle:

- ▶ Position the touch probe at a position near the first touch point outside of the circle
- ▶ Select the probe direction by soft key
- ▶ Probing: Press the machine START button. If you are not using the automatic probing routine, you need to repeat this procedure. After the third probing operation, you can have the TNC calculate the center (four touch points are recommended)
- ▶ Terminate the probing procedure and switch to the evaluation menu: Press the EVALUATE soft key
- ▶ **Datum:** Enter the coordinates of the datum and confirm your entry with the **SET DATUM** soft key, or write the values to a table (see "Writing measured values from the touch probe cycles in a datum table", page 551, or see "Writing measured values from the touch probe cycles in the preset table", page 552)
- ▶ To terminate the probe function, press the **END** soft key

After the probing procedure is completed, the TNC displays the current coordinates of the circle center and the circle radius PR.

Setting the datum using multiple holes/cylindrical studs

A second soft-key row provides a soft key for using multiple holes or cylindrical studs to set the datum. You can set the intersection of two or more elements as datum.

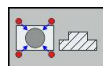
Select the probing function for the intersection of holes/cylindrical studs:



- ▶ Select the touch probe function: Press the **PROBING CC** soft key



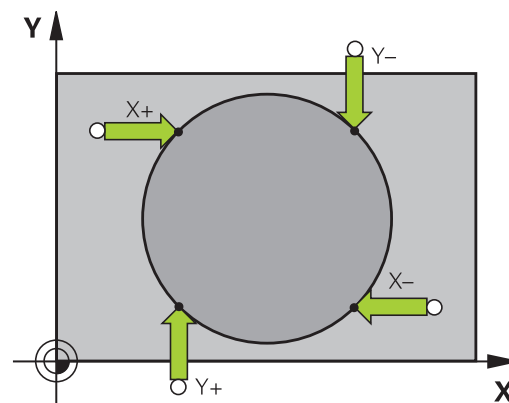
- ▶ Hole is to be probed automatically:
Define by soft key



- ▶ Circular stud is to be probed automatically:
Define by soft key

Preposition the touch probe approximately in the center of the hole or near the first touch point of the circular stud. After you have pressed the NC Start key, the TNC automatically probes the points on the circle.

Move the touch probe to the next hole, repeat the probing operation and have the TNC repeat the probing procedure until all the holes have been probed to set the datum.

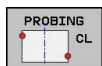


Setting the datum in the intersection of multiple holes:



- ▶ Preposition the touch probe approximately in the center of the hole
- ▶ Hole is to be probed automatically: Define by soft key
- ▶ To probe the workpiece, press the machine START button. The touch probe probes the circle automatically.
- ▶ Repeat the probing procedure for the remaining elements
- ▶ Terminate the probing procedure and switch to the evaluation menu: Press the **EVALUATE** soft key
- ▶ **Datum:** In the menu window, enter both coordinates of the circle center, confirm with the **SET DATUM** soft key, or write the values to a table (see "Writing measured values from the touch probe cycles in a datum table", page 551, or see "Writing measured values from the touch probe cycles in the preset table", page 552)
- ▶ Terminate the probing function: Press the **END** soft key

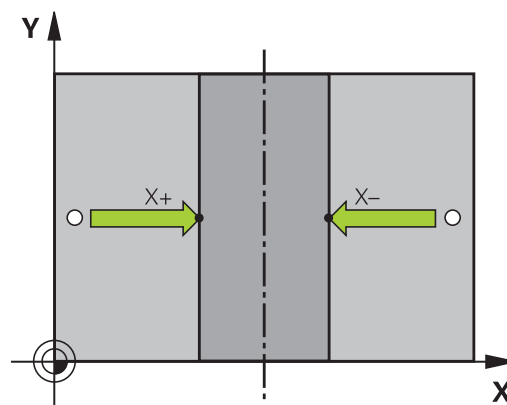
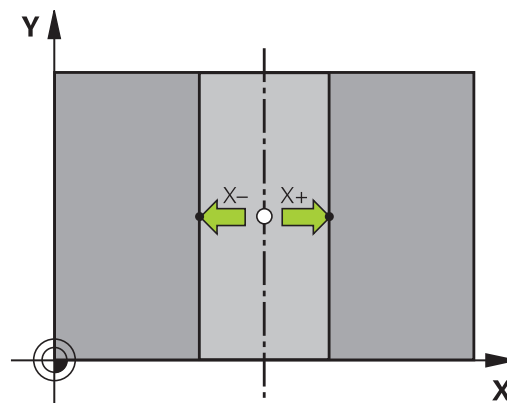
Setting a center line as datum



- ▶ Select the probing function: Press the **PROBING CL** soft key
- ▶ Position the touch probe at a position near the first touch point
- ▶ Select the probing direction by soft key
- ▶ Start the probing procedure: Press the NC Start button
- ▶ Position the touch probe at a position near the second touch point
- ▶ Start the probing procedure: Press the NC Start button
- ▶ **Datum:** Enter the coordinate of the datum in the menu window, confirm with the **SET DATUM** soft key, or write the value to a table (see "Writing measured values from the touch probe cycles in a datum table", page 551, or see "Writing measured values from the touch probe cycles in the preset table", page 552).
- ▶ Terminate the probing function: Press the **END** soft key



After you have measured the second touch point, you can use the evaluation menu to change the direction of the centerline. You can choose by soft key whether the datum or zero point should be set in the reference axis, minor axis or tool axis. This can be necessary if, for example, you would like to save the measured position in the reference and minor axis.



Manual operation and setup

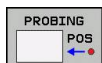
15.10 Datum setting with 3-D touch probe

Measuring workpieces with a 3-D touch probe

You can also use the touch probe in the **Manual Operation** and **El. Handwheel** operating modes to make simple measurements on the workpiece. Numerous programmable probe cycles are available for complex measuring tasks (see User's Manual for Cycles, Chapter 16, Automatic workpiece inspection). With a 3-D touch probe you can determine:

- Position coordinates, and from them,
- Dimensions and angles on the workpiece

Finding the coordinates of a position on an aligned workpiece



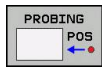
- ▶ Select the probing function: Press the **PROBING POS** soft key
- ▶ Move the touch probe to a position near the touch point
- ▶ Select the probe direction and axis of the coordinate. Use the corresponding soft keys for selection
- ▶ Start the probing procedure: Press the machine START button

The TNC shows the coordinates of the touch point as reference point.

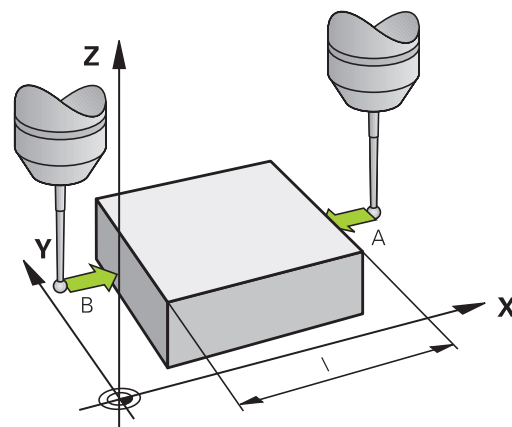
Finding the coordinates of a corner in the working plane

Find the coordinates of the corner point: see "Corner as datum ", page 564. The TNC displays the coordinates of the probed corner as reference point.

Measuring workpiece dimensions



- ▶ Select the probing function: Press the **PROBING POS** soft key
- ▶ Position the touch probe at a position near the first touch point A
- ▶ Select the probing direction by soft key
- ▶ Start the probing procedure: Press the machine START button
- ▶ If you need the current datum later, write down the value that appears in the Datum display
- ▶ Datum: Enter "0"
- ▶ Cancel the dialog: Press the **END** key
- ▶ Select the probing function again: Press the **PROBING POS** soft key
- ▶ Position the touch probe at a position near the second touch point B
- ▶ Select the probe direction with the soft keys: Same axis but from the opposite direction
- ▶ Start the probing procedure: Press the machine START button



The value displayed as datum is the distance between the two points on the coordinate axis.

To return to the datum that was active before the length measurement:

- ▶ Select the probing function: Press the **PROBING POS** soft key
- ▶ Probe the first touch point again
- ▶ Set the datum to the value that you wrote down previously
- ▶ Cancel the dialog: Press the **END** key

Measuring angles

You can use the 3-D touch probe to measure angles in the working plane. You can measure

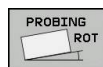
- the angle between the angle reference axis and a workpiece edge, or
- the angle between two sides

The measured angle is displayed as a value of maximum 90°.

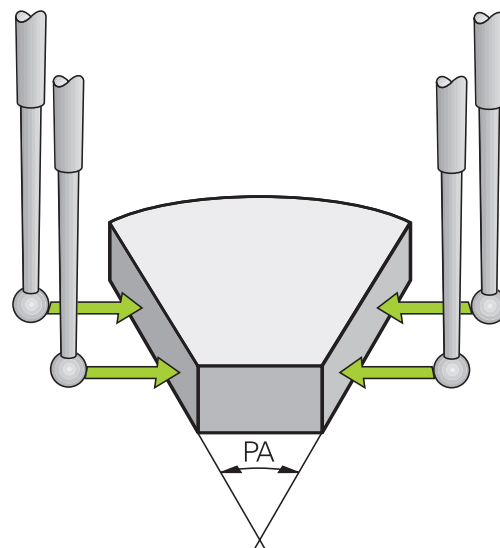
Manual operation and setup

15.10 Datum setting with 3-D touch probe

Finding the angle between the angle reference axis and a workpiece edge

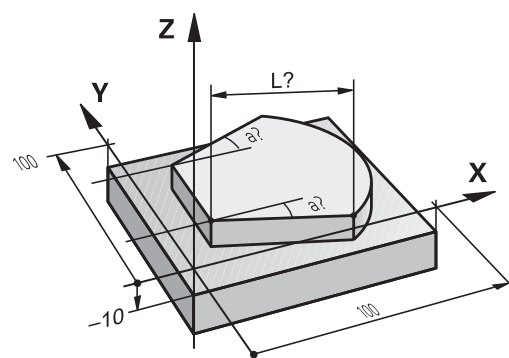


- ▶ Select the probe function by pressing the **PROBING ROT** soft key
- ▶ Rotation angle: If you will need the current basic rotation later, write down the value that appears under Rotation angle
- ▶ Make a basic rotation with workpiece edge to be compared see "Compensating workpiece misalignment with 3-D touch probe ", page 559
- ▶ Press the **PROBING ROT** soft key to display the angle between the angle reference axis and the workpiece edge as the rotation angle
- ▶ Cancel the basic rotation, or restore the previous basic rotation
- ▶ Set the rotation angle to the value that you previously wrote down



Measuring the angle between two workpiece edges

- ▶ Select the probe function by pressing the **PROBING ROT** soft key
- ▶ Rotation angle: If you need the current basic rotation later, write down the displayed rotation angle
- ▶ Make a basic rotation with first workpiece edge see "Compensating workpiece misalignment with 3-D touch probe ", page 559
- ▶ Probe the second edge as for a basic rotation, but do not set the rotation angle to zero!
- ▶ Press the **PROBING ROT** soft key to display the angle PA between the workpiece edges as the rotation angle
- ▶ Cancel the basic rotation, or restore the previous basic rotation by setting the rotation angle to the value that you wrote down previously

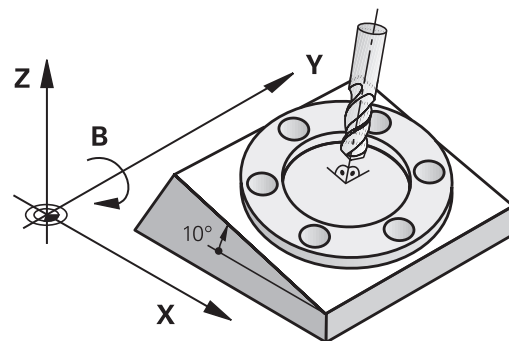


15.11 Tilting the working plane (Option #8)

Application, function



The functions for tilting the working plane are interfaced to the TNC and the machine tool by the machine tool builder. With some swivel heads and tilting tables, the machine tool builder determines whether the entered angles are interpreted as coordinates of the rotary axes or as angular components of a tilted plane. Refer to your machine manual.



The TNC supports the tilting functions on machine tools with swivel heads and/or tilting tables. Typical applications are, for example, oblique holes or contours in an oblique plane. The working plane is always tilted around the active datum. The program is written as usual in a main plane, such as the X/Y plane, but is executed in a plane that is tilted relative to the main plane.

There are three functions available for tilting the working plane:

- Manual tilting with the **3-D ROT** soft key in the Manual Operation mode and Electronic Handwheel mode, see "To activate manual tilting:", page 574
- Tilting under program control, Cycle **19 WORKING PLANE** in the part program (see User's Manual for Cycles, Cycle 19 WORKING PLANE)
- Tilting under program control, **PLANE** function in the part program see "The PLANE Function: Tilting the working plane (Software Option 8)", page 437

The TNC functions for "tilting the working plane" are coordinate transformations. The working plane is always perpendicular to the direction of the tool axis.

Manual operation and setup

15.11 Tilting the working plane (Option #8)

When tilting the working plane, the TNC differentiates between two machine types:

■ Machine with tilting table

- You must tilt the workpiece into the desired position for machining by positioning the tilting table, for example with an L block.
- The position of the transformed tool axis **does not change** in relation to the machine-based coordinate system. Thus if you rotate the table—and therefore the workpiece—by 90° for example, the coordinate system **does not rotate**. If you press the Z+ axis direction button in the Manual Operation mode, the tool moves in Z+ direction.
- In calculating the transformed coordinate system, the TNC considers only the mechanically influenced offsets of the particular tilting table (the so-called "translational" components).

■ Machine with swivel head

- You must tilt the workpiece into the desired position for machining by positioning the swivel head, for example with an L block
- The position of the transformed tool axis changes in relation to the machine-based coordinate system. Thus if you rotate the swivel head of your machine—and therefore the tool—in the B axis by 90° for example, the coordinate system also rotates. If you press the Z+ axis direction button in the Manual Operation mode, the tool moves in X+ direction of the machine-based coordinate system.
- In calculating the transformed coordinate system, the TNC considers both the mechanically influenced offsets of the particular swivel head (the so-called "translational" components) and offsets caused by tilting of the tool (3-D tool length compensation).



The TNC only supports tilting the working plane with spindle axis Z.

Traversing reference points in tilted axes

The TNC automatically activates the tilted working plane if this function was enabled when the control was switched off. Then the TNC moves the axes in the tilted coordinate system when an axis-direction key is pressed. Position the tool in such a way that a collision is excluded during the subsequent crossing of the reference points. To cross the reference points you have to deactivate the "Tilt Working Plane" function, see "To activate manual tilting:", page 574.



Danger of collision!

Be sure that the function for tilting the working plane is active in the Manual Operation mode and that the angle values entered in the menu match the actual angles of the tilted axis.

Deactivate the "Tilt Working Plane" function before you cross the reference points. Take care that there is no collision. Retract the tool from the current position first, if necessary.

Position display in a tilted system

The positions displayed in the status window (**ACTL.** and **NOML.**) are referenced to the tilted coordinate system.

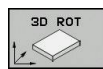
Limitations on working with the tilting function

- The actual-position-capture function is not allowed if the tilted working plane function is active.
- PLC positioning (determined by the machine tool builder) is not possible.

Manual operation and setup

15.11 Tilting the working plane (Option #8)

To activate manual tilting:



- To select manual tilting, press the **3-D ROT** soft key.



- Use the arrow keys to move the highlight to the **Manual Operation** menu item



- To activate manual tilting, press the **ACTIVE** soft key

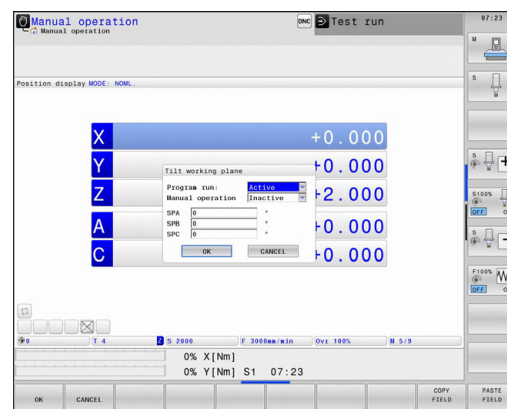


- Use the arrow keys to position the highlight on the desired rotary axis

- Enter the tilt angle



- To conclude entry, press the **END** key



If the tilted working plane function is active and the TNC moves the machine axes in accordance with the tilted axes, the status display shows the symbol.

If you activate the "Tilt working plane" function for the Program Run operating mode, the tilt angle entered in the menu becomes active in the first block of the part program. If you use Cycle **19 WORKING PLANE** or the **PLANE** function in the part program, the angle values defined there are in effect. Angle values entered in the menu will be overwritten.

To deactivate manual tilting

To reset the tilting function, set the desired operating modes in the menu **Tilt working plane** to inactive.

A programmed **PLANE RESET** only resets tilting in the Program Run, not in manual operation.

Tilting the working plane (Option #8) 15.11

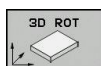
Setting the current tool-axis direction as the active machining direction



This function must be enabled by your machine manufacturer. Refer to your machine manual.

In the Manual Operation and El. Handwheel modes of operation you can use this function to move the tool via the external direction keys or with the handwheel in the direction that the tool axis is currently pointed. Use this function if

- You want to retract the tool in the direction of the tool axis during program interrupt of a 5-axis machining program.
- You want to machine with an inclined tool using the handwheel or the external direction keys in the Manual Operation mode.



- ▶ To select manual tilting, Press the 3-D ROT soft key



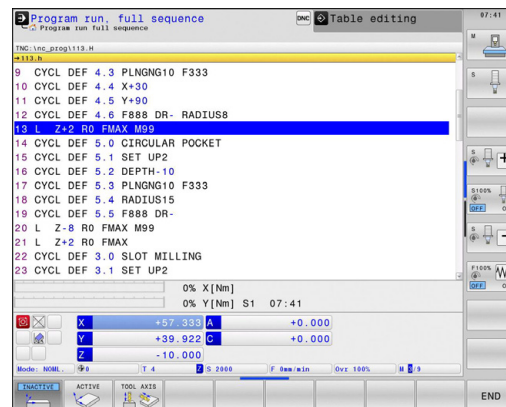
- ▶ Use the arrow keys to move the highlight to the **Manual Operation** menu item




- ▶ To activate the current tool-axis direction as the active machining direction, press the Tool Axis soft key



- ▶ To conclude entry, press the END key



To reset the tilting function, set the **Manual Operation** menu item in the "Tilt working plane" menu to inactive.

The  symbol appears in the status display when the **Move in tool-axis direction** function is active.



This function is even available when you interrupt program run and want to move the axes manually.

Manual operation and setup

15.11 Tilting the working plane (Option #8)

Setting the datum in a tilted coordinate system

After you have positioned the rotary axes, set the preset in the same manner as for a non-tilted system. The behavior of the TNC during datum setting depends on the setting in machine parameter

CfgPresetSettings/chkTiltingAxes:

- **chkTiltingAxes: On** With an active tilted working plane, the TNC checks during datum setting in the X, Y and Z axes whether the current coordinates of the rotary axes agree with the tilt angles that you defined (3-D ROT menu). If the tilted working plane function is not active, the TNC checks whether the rotary axes are at 0° (actual positions). If the positions do not agree, the TNC will display an error message.
- **chkTiltingAxes: Off** The TNC does not check whether the current coordinates of the rotary axes (actual positions) agree with the tilt angles that you defined.



Danger of collision!

Always set a reference point in all three reference axes.

16

**Positioning with
Manual Data Input**

16.1 Programming and executing simple machining operations

16.1 Programming and executing simple machining operations

The **Positioning with Manual Data Input** mode of operation is particularly convenient for simple machining operations or to pre-position the tool. It enables you to write a short program in HEIDENHAIN conversational programming or in ISO format, and execute it immediately. You can also call TNC cycles. The program is stored in the file \$MDI. In the **Positioning with MDI** mode of operation, the additional status display can also be activated.

Positioning with manual data input (MDI)



Limitation

The following functions are not available in the **Positioning with MDI** operating mode:

- FK free contour programming
- Program section repeats
- Subprogramming
- Path compensations RL and RR
- The programming graphics
- Program call **PGM CALL**
- The program-run graphics



- ▶ Select the **Positioning with MDI** mode of operation. Program the file \$MDI as you wish



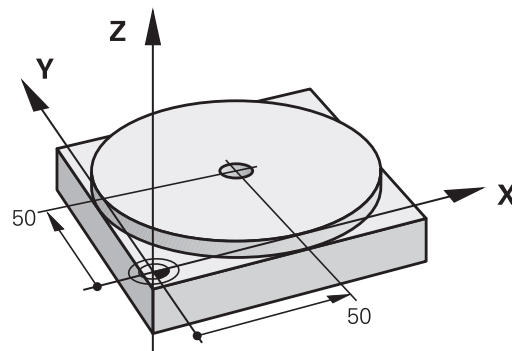
- ▶ To start program run, press the machine START button.

Programming and executing simple machining operations 16.1

Example 1

A hole with a depth of 20 mm is to be drilled into a single workpiece. After clamping and aligning the workpiece and setting the datum, you can program and execute the drilling operation in a few lines.

First you pre-position the tool with straight-line blocks to the hole center coordinates at a setup clearance of 5 mm above the workpiece surface. Then drill the hole with Cycle **200 DRILLING**.



0 BEGIN PGM \$MDI MM	
1 TOOL CALL 1 Z S2000	Call the tool: tool axis Z, spindle speed 2000 rpm
2 L Z+200 R0 FMAX	Retract the tool (F MAX = rapid traverse)
3 L X+50 Y+50 R0 FMAX M3	Move the tool at F MAX to a position above the hole, spindle on
4 CYCL DEF 200	Define the DRILLING cycle
Q200=5 ;SET-UP CLEARANCE	Set-up clearance of the tool above the hole
Q201=-15 ;DEPTH	Hole depth (algebraic sign=working direction)
Q206=250 ;FEED RATE FOR PLNGNG	Feed rate for drilling
Q202=5 ;	Depth of each infeed before retraction
Q210=0 ;DWELL TIME AT TOP	Dwell time after every retraction in seconds
Q203=-10 ;SURFACE COORDINATE	Coordinate of the workpiece surface
Q204=20 ;2ND SET-UP CLEARANCE	Set-up clearance of the tool above the hole
Q211=0.2 ;DWELL TIME AT DEPTH	Dwell time in seconds at the hole bottom
5 CYCL CALL	Call the DRILLING cycle
6 L Z+200 R0 FMAX M2	Retract the tool
7 END PGM \$MDI MM	End of program

Straight-line function: see "Straight line L", page 223

DRILLING cycle: See User's Manual, Cycles, Cycle 200 DRILLING.

16

Positioning with Manual Data Input

16.1 Programming and executing simple machining operations

Example 2: Correcting workpiece misalignment on machines with rotary tables

- ▶ Use the 3-D touch probe to rotate the coordinate system, "Compensating workpiece misalignment with 3-D touch probe "
- ▶ Write down the rotation angle and cancel the basic rotation



- ▶ Select operating mode: **Positioning with MDI**



- ▶ Select the rotary table axis, enter the rotation angle and feed rate you wrote down, e.g. **L C +2.561 F50**



- ▶ Conclude entry



- ▶ Press the machine START button: The rotation of the table corrects the misalignment

Protecting and erasing programs in \$MDI

The \$MDI file is generally intended for short programs that are only needed temporarily. Nevertheless, you can store a program, if necessary, by proceeding as described below:



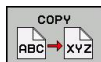
- ▶ Select the **Programming** mode of operation



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



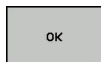
- ▶ Move the highlight to the **\$MD** file



- ▶ Copy a file: Press the **COPY** soft key

DESTINATION FILE =

- ▶ Enter the name under which you want to save the current contents of the \$MDI file, e.g. **BORE**.



- ▶ Press the OK soft key



- ▶ Close the file manager: END soft key

For more information: see "Copying a single file", page 121.

17

**Test run and
program run**

Test run and program run

17.1 Graphics

17.1 Graphics

Application

In the **Program Run, Single Block** and **Program Run, Full Sequence** operating modes as well as in the **Test Run** operating mode, the TNC simulates the machining of the workpiece.

The TNC features the following views:

- Plan view
- Projection in three planes
- 3-D view



In the **Test Run** operating mode, you can also use the 3-D line graphics.

The TNC graphic depicts the workpiece as if it were being machined with a cylindrical end mill.

If a tool table is active, the TNC also considers the entries in the LCUTS, T-ANGLE and R2 columns.

With the model type 3-D **graphic setting** and in turning mode you also see the indexable inserts of the turning tools from **toolturn.trn**.

The TNC will not show a graphic if

- the current program has no valid workpiece blank definition
- no program is selected
- if the BLK FORM block was not yet executed during the workpiece blank definition with the aid of a subprogram



The simulation of programs with 5-axis machining or tilted machining might run at reduced speed. With the MOD menu **Graphic settings** you can decrease the **model quality** and in that way increase the speed of simulation.

Graphics without Option #20 Advanced graphic features

Without Option #20, no model is available in **Program Run, Single Block** and **Program Run, Full Sequence** operating modes and the **Test Run** operating mode.

The **PROGRAM + GRAPHICS** and **GRAPHICS** soft keys are dimmed.





The line graphic in **Programming** operating mode also functions without Option #20.

Speed of the Setting test runs



The most recently set speed stays active until a power interruption. After the control is switched on the speed is set to FMAX.

After you have started a program, the TNC displays the following soft keys with which you can set the simulation speed:

Soft key	Functions
	Perform the test run at the same speed at which the program will be run (programmed feed rates are taken into account)
	Increase the simulation speed incrementally
	Decrease the simulation speed incrementally
	Test run at the maximum possible speed (default setting)

You can also set the simulation speed before you start a program:



- ▶ Select the function for setting the simulation speed




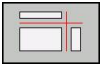
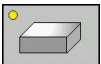
- ▶ Select the desired function by soft key, e.g. incrementally increasing the simulation speed

Test run and program run

17.1 Graphics

Overview: Display modes




In the **Program Run, Single Block** and **Program Run, Full Sequence** operating modes as well as in the **Test Run** operating mode, the TNC displays the following soft keys:

Soft key	View
	Plan view
	Projection in three planes
	3-D view



The position of the soft keys depends on the selected operating mode.

The **Test Run** operating mode additionally offers the following views:

Soft key	View
	Volume view
	Volume view and tool paths
	Tool paths

Limitations during program run



The result of the simulation can be faulty if the TNC's computer is overloaded with complicated processing tasks.

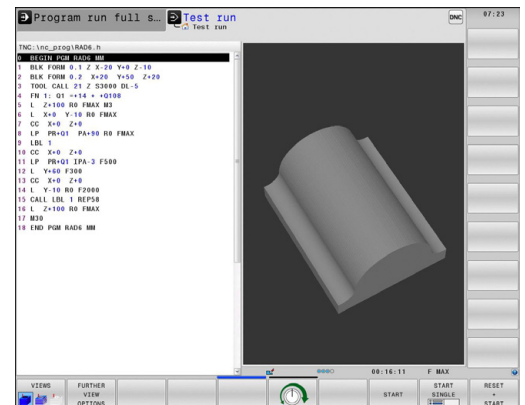
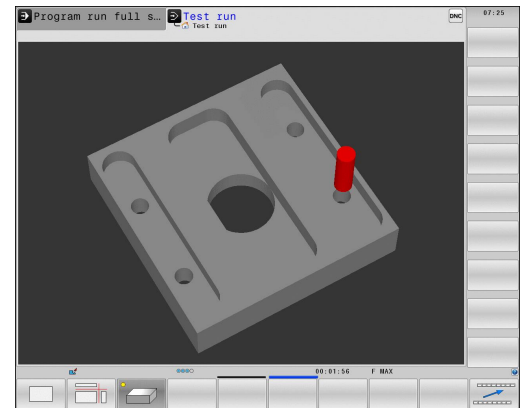
3-D view

Choose 3-D view:

The high-resolution 3-D view enables you to display the surface of the machined workpiece in greater detail. With a simulated light source, the TNC creates realistic light and shadow conditions.



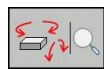
- Press the 3-D view soft key



Test run and program run

17.1 Graphics

Rotating, enlarging, reducing and shifting the 3-D view



- Select the functions for rotating and enlarging/reducing: The TNC displays the following soft keys:

Soft keys	Function
	Rotate in 5° steps about the vertical axis
	Tilt in 5° steps about the horizontal axis
	Enlarge the graphic stepwise
	Reduce the graphic stepwise
	Reset the graphic to its original size and angle
	► Shift the soft-key row




Soft keys	Function
	Move the graphic upward or downward
	Move the graphic to the left or right
	Reset the graphic to its original position and angle

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




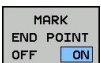
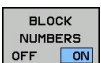
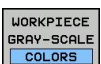
- In order to rotate the model shown in three dimensions you hold the right mouse button down and move the mouse. If you simultaneously press the shift key, you can only rotate the model horizontally or vertically.
- To shift the model shown: Hold the center mouse button or the wheel button down and move the mouse. If you simultaneously press the shift key, you can only move 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 TNC zooms in on the defined area.
- To rapidly magnify or reduce any area: Rotate the mouse wheel backwards or forwards.
- To return to the standard display: Press the shift key and simultaneously double-click with the right mouse key. The rotation angle is maintained if you only double-click with the right mouse key.

3-D view in the Test Run mode of operation

The **Test Run** operating mode additionally offers the following views:

Soft keys	Function
	Volume view
	Volume view and tool paths
	Tool paths

The **Test Run** operating mode additionally offers the following functions:

Soft keys	Function
	Show workpiece blank frame
	Highlight workpiece edges
	Show a transparent workpiece
	Show the end points of the tool paths
	Show the block numbers of the tool paths
	Show the workpiece in color

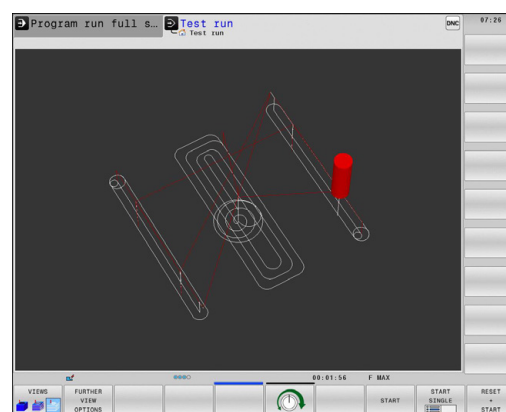
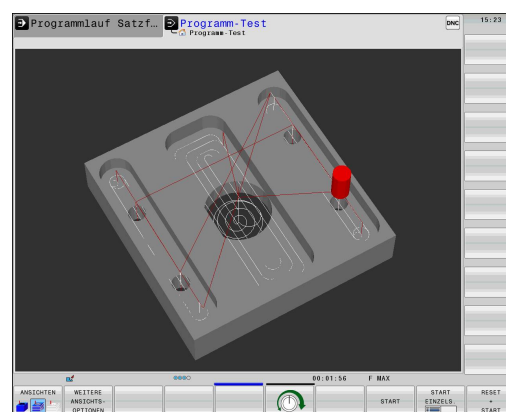
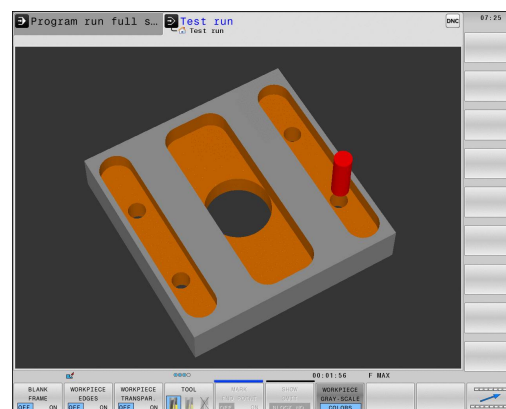


Note that the range of functions depends on the model quality selected. You can select the model quality in the MOD function **Graphic settings**.



By showing the tool paths you can depict the programmed paths of the TNC in three dimensions. A powerful zoom function is available for recognizing details quickly.

In particular, you can use the tool paths display to inspect programs created externally for irregularities before machining. This can help you to avoid undesirable traces of the machining process on the workpiece. Such traces of machining can occur when points are output incorrectly by the postprocessor. The TNC shows traverse movements in rapid traverse in red.



Test run and program run

17.1 Graphics

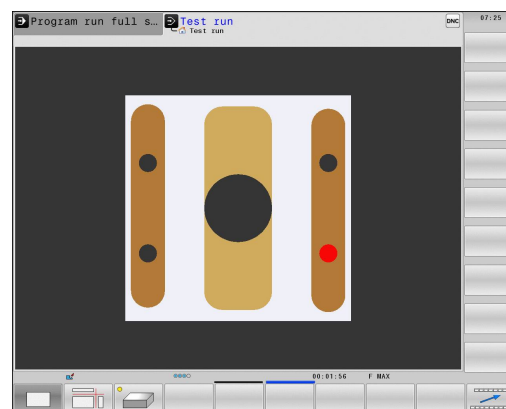
Plan view

Select the plan view in the **Test Run** operating mode:

- ▶ Press the **FURTHER VIEW OPTIONS** soft key
- ▶ Press the plan-view soft key

Select the plan view in the **Program Run, Single Block** and **Program Run, Full Sequence** operating modes:

- ▶ Press the **GRAPHICS** soft key
- ▶ Press the plan-view soft key



Projection in three planes

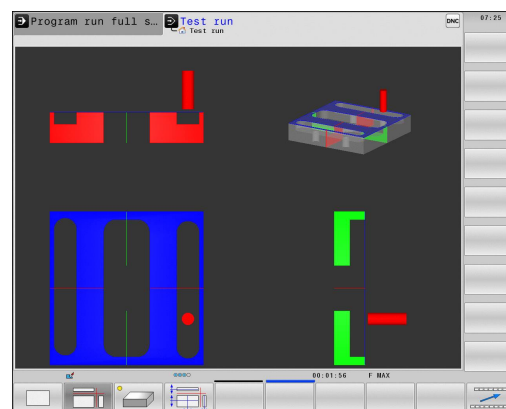
The simulation shows three sectional planes and a 3-D model, Similar to a technical drawing.

Select projection in three planes in the **Test Run** operating mode:

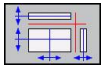
- ▶ Press the **FURTHER VIEW OPTIONS** soft key
- ▶ Press the view-in-three-planes soft key

Select projection in three planes in the **Program Run, Single Block** and **Program Run, Full Sequence** operating modes:

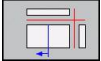
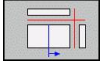


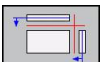
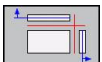
- ▶ Press the **FURTHER VIEW OPTIONS** soft key
- ▶ Press the view-in-three-planes soft key



Move the sectional planes



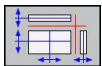
- Select the functions for shifting the sectional plane. The TNC offers the following soft keys:

Soft keys	Function
 	Shift the vertical sectional plane to the right or left
 	Shift the vertical sectional plane forward or backward
 	Shift the horizontal sectional plane upwards or downwards

The position of the sectional planes is visible during shifting.

The default setting of the sectional plane is selected so that it lies in the working plane in the workpiece center and in the tool axis on the top surface.

Return sectional planes to default setting:




- Select the function for resetting the sectional planes.

Test run and program run

17.1 Graphics



Repeating graphic simulation

A part program can be graphically simulated as often as desired. To do so you can reset the graphic to the workpiece blank.

Soft key	Function
	Show the unmachined workpiece blank

Tool display

Regardless of the operating mode, you can also show the tool during the simulation.

Soft key	Function
	Program Run, Full Sequence / Program Run, Single Block
	Test Run

Measurement of machining time

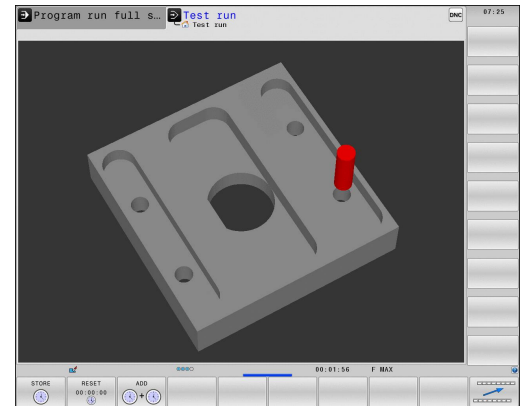
Machining time in the Test Run mode of operation

The control calculates the duration of the tool movements and displays this as machining time in the test run. The control takes feed movements and dwell times into account.

The time calculated by the control can only conditionally be used for calculating the production time because the control does not account for machine-dependent times, such as tool change.



The machining times shown for milling/turning operations in the simulation do not correspond to the actual machining times.



Machining time in the machine operating modes

Time display from program start to program end. The timer stops whenever machining is interrupted.

Activating the stopwatch function



- Shift the soft-key row until the soft-key for the stopwatch functions appears



- Select the stopwatch functions



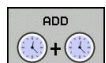
- Select the desired function via soft key, e.g. saving the displayed time

Soft key

Stopwatch functions



Store displayed time



Display the sum of stored time and displayed time



Clear displayed time

Test run and program run

17.2 Showing the workpiece blank in the working space

17.2 Showing the workpiece blank in the working space

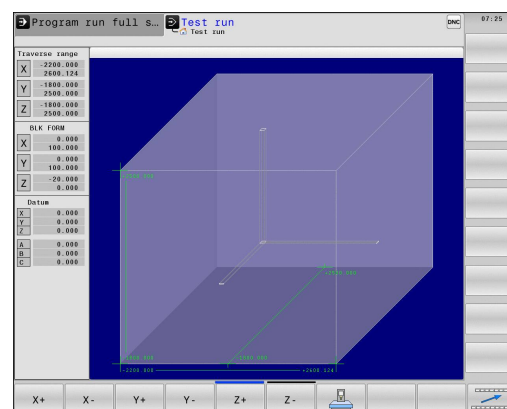
Application

In the **Test Run** operating mode, you can graphically check the position of the workpiece blank or reference point in the machine's working space and activate work space monitoring in the **Test Run** mode: Press the **BLANK IN WORK SPACE** soft key to activate this function. You can use the soft key **SW LIMIT MONITORING** (2nd soft-key row) to activate or deactivate the function.

A transparent cuboid represents the workpiece blank. Its dimensions are shown in the **BLK FORM** table. The TNC takes the dimensions from the workpiece blank definition of the selected program. The workpiece cuboid defines the coordinate system. Its datum lies within the traverse-range cuboid.

For a test run it normally does not matter where the workpiece blank is located within the working space. However, if you activate working-space monitoring, you must graphically shift the workpiece blank so that it lies within the working space. Use the soft keys shown in the table.

You can also activate the current datum for the **Test Run** operating mode (see the following table).



Soft keys	Function
<div>X+</div> <div>X-</div>	Shift workpiece blank in positive/negative X direction
<div>Y+</div> <div>Y-</div>	Shift workpiece blank in positive/negative Y direction
<div>Z+</div> <div>Z-</div>	Shift workpiece blank in positive/negative Z direction
<div> </div>	Show workpiece blank referenced to the set datum
<div>SW limit monitoring</div>	Switch monitoring function on or off







Note that even with **BLK FORM CYLINDER**, a cuboid is shown in the working space as workpiece blank. When **BLK FORM ROTATION** is used, no workpiece blank is shown in the working space.

17.3 Functions for program display

Overview

In the **Program Run, Single Block** and **Program Run, Full Sequence** modes of operation, the TNC displays the following soft keys for displaying a part program in pages:

Soft key	Functions
	Go back in the program by one screen
	Go forward in the program by one screen
	Go to the start of the program
	Go to the end of the program

Test run and program run

17.4 Test Run

17.4 Test Run

Application

In the **Test Run** mode of operation, you can simulate programs and program sections to reduce programming errors during program run. The TNC checks the programs for the following:

- Geometrical incompatibilities
- Missing data
- Impossible jumps
- Violation of the machine's working space

The following functions are also available:

- Blockwise test run
- Interruption of test at any block
- Optional block skip
- Functions for graphic simulation
- Measure machining time
- Additional status display



Danger of collision!

The TNC cannot graphically simulate all traverse motions actually performed by the machine. These include

- Traverse motions during tool change, if the machine manufacturer defined them in a tool-change macro or via the PLC
- Positioning movements that the machine manufacturer defined in an M-function macro
- Positioning movements that the machine manufacturer performs via the PLC

HEIDENHAIN therefore recommends proceeding with caution for every new program, even when the program test did not output any error message, and no visible damage to the workpiece occurred.

After a tool call, the TNC always starts a program test at the following position:

- In the working plane in the center of the defined **BLK FORM**
- In the tool axis, 1 mm above the **MAX** point defined in the **BLK FORM**

With rotationally symmetric workpiece blanks, the TNC starts a program test run after a tool call at the following position:

- In the machining plane at the position X=0, Y=0
- In the tool axis 1 mm above the defined workpiece blank

In order to ensure unambiguous behavior during program run, after a tool change you should always move to a position from which the TNC can position the tool for machining without causing a collision.



Your machine tool builder can also define a tool-change macro for the **Test Run** operating mode. This macro will simulate the exact behavior of the machine. Refer to your machine manual.

Execute test run



If the central tool file is active, a tool table must be active (status S) to conduct a test run. Select a tool table via the file manager in the **Test Run** mode of operation.

For turning tools you can select a turning tool table with the file extension .trn, compatible with the selected tool table. This means that the turning tools must match in both selected tables.

You can select any preset table (status S) for the test run.

After **RESET + START**, line 0 of the temporarily loaded preset table automatically displays the momentarily active datum from **Preset.pr** (execution). Line 0 is selected when starting the test run until you define another datum in the NC program. All datums from lines > 0 are read by the control from the selected preset table of the test run.

With the **BLANK IN WORK SPACE** function, you activate a workspace monitor for the test run, see "Showing the workpiece blank in the working space", page 594.







- Select the **Test Run** operating mode



- Call the file manager with the **PGM MGT** key and select the file you wish to test

The TNC then displays the following soft keys:

Soft key	Functions
	Reset the blank form and test the entire program
	Test the entire program
	Test each program block individually
	Halt test run (soft key only appears once you have started the test run)

You can interrupt the test run and continue it again at any point — even within a fixed cycle. In order to continue the test, the following actions must not be performed:

- Selecting another block with the arrow keys or the **GOTO** key
- Making changes to the program
- Selecting a new program

Test run and program run

17.5 Program run

17.5 Program run

Application

In the **Program Run, Full Sequence** mode of operation the TNC executes a part program continuously to its end or up to a program stop.

In the **Program Run, Single Block** mode of operation you must start each block separately by pressing the machine **START** button. With point pattern cycles and **CYCL CALL PAT**, the control stops after each point.

You can use the following TNC functions in the **Program Run, Single Block** and **Program Run, Full Sequence** operating modes:

- Interrupt program run
- Start the program run from a certain block
- Optional block skip
- Edit the tool table TOOL.T
- Checking and changing Q parameters
- Superimposing handwheel positioning
- Functions for graphic simulation
- Additional status display

Running a part program

Preparation

- 1 Clamp the workpiece to the machine table.
- 2 Set the datum
- 3 Select the necessary tables and pallet files (status M)
- 4 Select the part program (status M)



You can adjust the feed rate and spindle speed with the override knobs.



It is possible to reduce the feed rate when starting the NC program using the **FMAX** soft key. The reduction applies to all rapid traverse and feed rate movements. The value you enter is no longer in effect after the machine has been turned off and on again. In order to re-establish the respectively defined maximum feed rate after switch-on, you need to re-enter the corresponding value.

The behavior of this function varies depending on the respective machine. Refer to your machine manual.

Program Run, Full Sequence

- Start the part program with the machine **START** button

Program Run, Single Block

- Start each block of the part program individually with the machine **START** button

Test run and program run

17.5 Program run

Interrupt machining

There are several ways to interrupt a program run:

- Programmed interruptions
- Pressing the machine **STOP** button
- Switching to **Program Run, Single Block** mode

If the TNC registers an error during program run, it automatically interrupts the machining process.

Programmed interruptions

You can define interruptions directly in the part program. The TNC interrupts the program run at a block containing one of the following entries:

- **STOP** (with and without miscellaneous function)
- Miscellaneous function **M0**, **M2** or **M30**
- Miscellaneous function **M6** (determined by the machine tool builder)

Interruption through the machine **STOP** button

- ▶ Press the machine **STOP** button: The block that the TNC is currently executing is not completed. The NC stop signal in the status display blinks (see table)
- ▶ If you do not wish to continue the machining process, you can reset the TNC with the **INTERNAL STOP** soft key. The NC stop signal in the status display goes out. In this case, the program must be restarted from the program beginning

Icon	Meaning
	Program run is stopped

Interruption of machining by switching to the Program Run, Single Block mode of operation

You can interrupt a program that is being run in the **Program Run, Full Sequence** mode of operation by switching to the **Program Run, Single Block mode**. The TNC interrupts the machining process at the end of the current block.

Moving the machine axes during an interruption

You can move the machine axes during an interruption in the same way as in the **Manual Operation** mode.



Danger of collision!

If you interrupt program run while the working plane is tilted, you can switch the coordinate system between tilted and non-tilted, as well as to the active tool axis direction, by pressing the **3-D ROT** soft key.

The functions of the axis direction buttons, the electronic handwheel and the positioning logic for returning to the contour are then evaluated by the TNC. When retracting the tool make sure the correct coordinate system is active and the angular values of the tilt axes are entered in the 3-D ROT menu, if necessary.

Example:

Retracting the spindle after tool breakage

- ▶ Interrupt machining
- ▶ Enable the external direction keys: Press the **MANUAL TRAVERSE** soft key
- ▶ Move the axes with the machine axis direction buttons.



On some machines you may have to press the machine **START** button after the **MANUAL OPERATION** soft key to enable the axis direction buttons. Refer to your machine manual.

Resuming program run after an interruption



If you cancel a program with INTERNAL STOP, you have to start the program with the **RESTORE POS. AT N** function or with GOTO "0".

If a program run is interrupted during a fixed cycle, the program must be resumed from the beginning of the cycle. This means that some machining operations will be repeated.

If you interrupt a program run during execution of a subprogram or program section repeat, use the **RESTORE POS AT N** function to return to the position at which the program run was interrupted.

Test run and program run

17.5 Program run

When a program run is interrupted, the TNC stores:

- The data of the last defined tool
- Active coordinate transformations (e.g. datum shift, rotation, mirroring)
- The coordinates of the circle center that was last defined



Note that the stored data remain active until they are reset (e.g. if you select a new program).

The stored data are used for returning the tool to the contour after manual machine axis positioning during an interruption (**RESTORE POSITION** soft key).

Resuming program run with the **START** button

You can resume program run by pressing the machine **START** button if the program was interrupted in one of the following ways:

- Machine **STOP** button pressed
- Programmed interruption

Resuming program run after an error

With an erasable error message:

- ▶ Remove the cause of the error
- ▶ Clear the error message from the screen: Press the **CE** key
- ▶ Restart the program, or resume program run where it was interrupted

With an non-erasable error message

- ▶ Press and hold the **END** key for two seconds. This induces a TNC system restart
- ▶ Remove the cause of the error
- ▶ Restart

If you cannot correct the error, write down the error message and contact your service agency.

Retraction after a power interruption



The **Retraction** mode of operation must be enabled and adapted by the machine tool builder. Refer to your machine manual.

With the **Retraction** mode of operation you can disengage the tool from the workpiece after an interruption in power.

The **Retraction** mode of operation is selectable in the following conditions:

- Power interruption
- Relay external DC voltage missing
- Traverse reference points

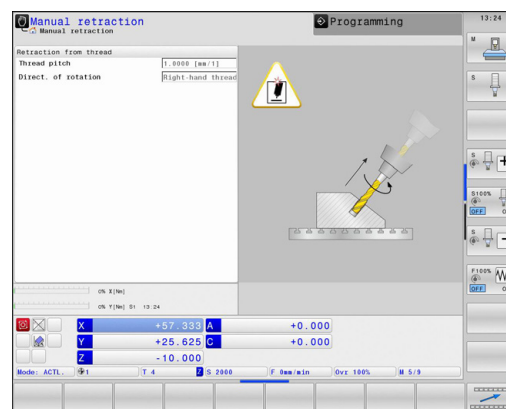
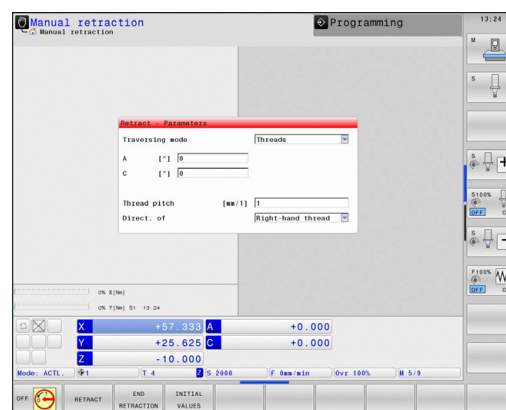
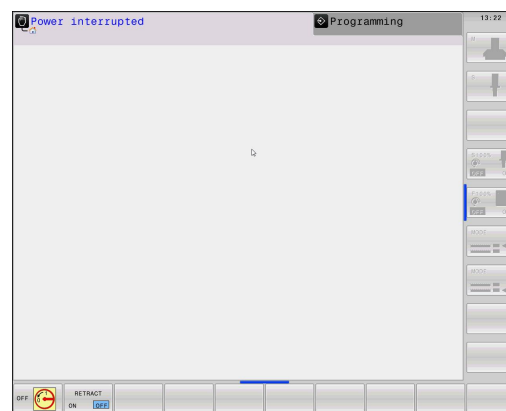
The **Retraction** operating mode offers the following modes of traverse:

Mode	Function
Machine axes	Movement of all axes in the original coordinate system
Tilted system	Movement of all axes in the active coordinate system Effective parameters: Position of the tilting axes
Tool axis	Movements of the tool axis in the active coordinate system
Thread	Movements of the tool axis in the active coordinate system with compensating movement of the spindle Effective parameters: Thread pitch and direction of rotation



The **tilted system** mode of traverse is available only when "tilting the working plane" (Option #8) is enabled on your TNC.

The TNC selects the mode of traverse and the associated parameters automatically. If the traverse mode or the parameters were not correctly chosen, you can change them manually.



17.5 Program run

**Danger of collision!**

For nonreferenced axes, the TNC adopts the most recently saved axis values. These values generally are not the exact actual axis positions!

As a result, for example, the tool might not move exactly along the actual tool direction. If the tool is still in contact with the workpiece, it can cause stress or damage to the tool and workpiece. Stress or damage to the workpiece or tool can also be caused by uncontrolled coasting or braking of axes after a power interruption. Move the axes carefully if the tool is still in contact with the workpiece. Set the feed rate override to the smallest values possible. If you use the handwheel, use a small feed rate factor.

The traverse range monitoring is not available for nonreferenced axes. Observe the axes while you move them. Do not move to the limits of traverse.

Example

The power failed while a thread cutting cycle in the tilted working plane was being performed. You have to retract the tap:

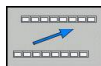
- ▶ Switch on the power supply for control and machine. The TNC starts the operating system. This process may take several minutes. Then the TNC will display the message "Power interrupted" in the screen header
-
- ▶ Activate the **Retraction** mode: Press the **RETRACT** soft key. The TNC displays the message **RETRACT**.
-
- ▶ Acknowledge the power interruption: Press the **CE** key. The TNC compiles the PLC program.
-
- ▶ Switch on the control voltage: The TNC checks the functioning of the EMERGENCY STOP circuit. If at least one axis is not referenced, you have to compare the displayed position values with the actual axis value and confirm their agreement. Follow the dialog, if required.
- ▶ Check the preselected traversing mode: if required, select **THREAD**
 - ▶ Check the preselected thread pitch: if required, enter the thread pitch
 - ▶ Check the preselected direction of rotation: if required, select the direction of thread rotation.
 Right-handed thread: The spindle turns in clockwise direction when moving into the workpiece and counterclockwise when retracting
 Left-hand thread: The spindle turns in counterclockwise direction when moving into the workpiece and clockwise when retracting

- ▶ Activate retraction: Press the **RETRACT** soft key

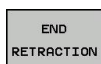
- ▶ Retraction: Retract the tool with the machine axis keys or the electronic handwheel

Axis key Z+: Retraction from the workpiece

Axis key Z-: Moving into the workpiece



- ▶ Exit retraction: Return to the original soft-key level



- ▶ End the **Retraction** mode: Press the **END RETRACTION** soft key. The TNC checks whether the **Retraction** mode can be ended. If necessary, follow the dialog.

- ▶ Answer the confirmation request: If the tool was not correctly retracted, press the **NO** soft key. If the tool was correctly retracted, press the **YES** soft key. The TNC hides the **retraction** dialog.
- ▶ Initialize the machine: if required, scan the reference points
- ▶ Establish the desired machine condition: if required, reset the tilted working plane

Any entry into program (mid-program startup)



The **RESTORE POS AT N** feature must be enabled and adapted by the machine tool builder. Refer to your machine manual.

With the **RESTORE POS AT N** feature (block scan) you can start a part program at any block you desire. The TNC scans the program blocks up to that point. Machining can be graphically simulated.

If you have interrupted a part program with an **INTERNAL STOP**, the TNC automatically offers the interrupted block N for mid-program startup.



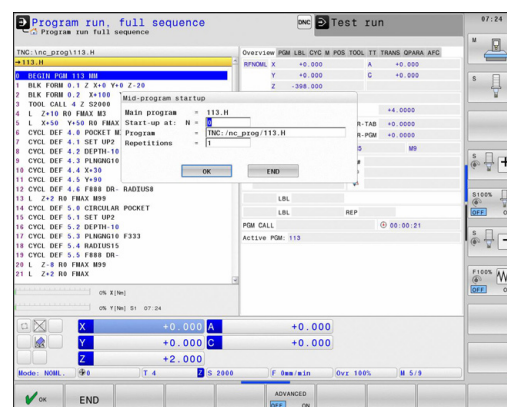
Mid-program startup must not begin in a subprogram.

All necessary programs, tables and pallet files must be selected in the **Program Run, Single Block** and **Program Run, Full Sequence** operating modes (status M).

If the program contains a programmed interruption before the startup block, the block scan is interrupted. Press the machine **START** button to continue the block scan.

After a block scan, return the tool to the calculated position with **RESTORE POSITION**.

Tool length compensation does not take effect until after the tool call and a following positioning block. This also applies if you have only changed the tool length.



17.5 Program run

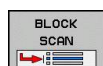


The TNC skips all touch probe cycles in a mid-program startup. Result parameters that are written to from these cycles might therefore remain empty.

You may not use mid-program startup if the following occurs after a tool change in the machining program:

- The program is started in an FK sequence
- The stretch filter is active
- Pallet management is used
- The program is started in a threading cycle (Cycles 17, 18, 19, 206, 207 and 209) or the subsequent program block
- Touch-probe cycles 0, 1 and 3 are used before program start

- ▶ Go to the first block of the current program to start a block scan:
Enter **GOTO "0"**



- ▶ Select mid-program startup: Press the **MID-PROGRAM STARTUP** soft key
- ▶ **Start-up at N:** Enter the block number N at which the block scan should end
- ▶ **Program:** Enter the name of the program containing block N
- ▶ **Repetitions:** If block N is located in a program section repeat or in a subprogram that is to be run repeatedly, enter the number of repetitions to be calculated in the block scan
- ▶ Start mid-program startup: Press the machine **START** button
- ▶ Contour approach (see following section)

Entering a program with the GOTO key



If you use the **GOTO** block number key for going into a program, neither the TNC nor the PLC will execute any functions that ensure a safe start.

If you use the GOTO block number key for going into a subprogram,

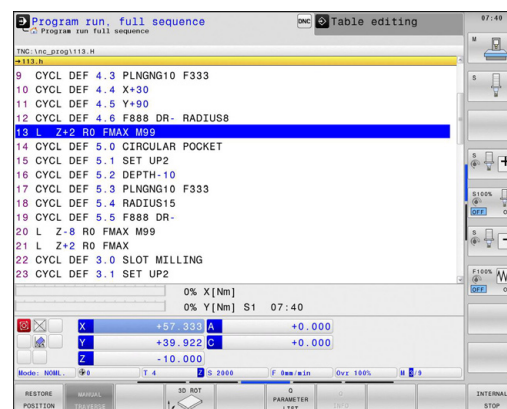
- the TNC will skip the end of the subprogram (**LBL 0**)
- the TNC will reset function M126 (Shorter-path traverse of rotary axes)

In such cases you must always use the mid-program startup function.

Returning to the contour

With the **RESTORE POSITION** function, the TNC returns to the workpiece contour in the following situations:

- Return to the contour after the machine axes were moved during a program interruption that was not performed with the **INTERNAL STOP** function
 - Return to the contour after a block scan with **RESTORE POS AT N**, for example after an interruption with **INTERNAL STOP**
 - Depending on the machine, if the position of an axis has changed after the control loop has been opened during a program interruption
- ▶ To select a return to contour, Press the **RESTORE POSITION** soft key
- ▶ Restore machine status, if required
- ▶ To move the axes in the sequence that the TNC suggests on the screen, press the machine **START** button, or
- ▶ To move the axes in any sequence: press the soft keys **RESTORE X**, **RESTORE Z**, etc., and activate each axis with the machine **START** button.
- ▶ To resume machining, press the machine **START** button.



Test run and program run

17.6 Automatic program start

17.6 Automatic program start

Application



The TNC must be specially prepared by the machine tool builder for use of the automatic program start function. Refer to your machine manual.



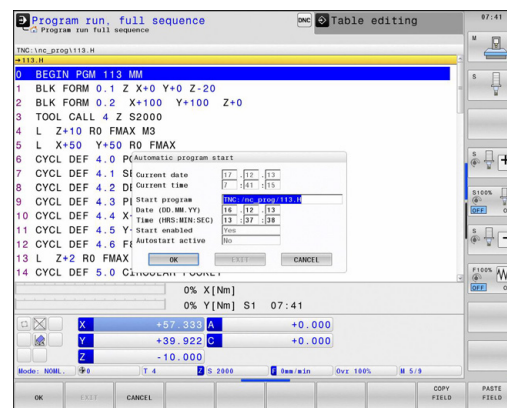
Caution: Danger for the operator!

The autostart function must not be used on machines that do not have an enclosed working space.

In a Program Run operating mode, you can use the **AUTOSTART** soft key (see figure at upper right) to define a specific time at which the program that is currently active in this operating mode is to be started:



- ▶ Show the window for entering the starting time (see figure at center right)
- ▶ **Time (hrs:min:sec):** Time of day at which the program is to be started
- ▶ **Date (DD.MM.YYYY):** Date on which the program is to be started
- ▶ To activate the start, press the **OK**



17.7 Optional block skip

Application

In a test run or program run, the control can skip over blocks that begin with a slash "/":



- ▶ To run or test the program without the blocks preceded by a slash, set the soft key to **ON**



- ▶ To run or test the program with the blocks preceded by a slash, set the soft key to **OFF**



This function does not work for **TOOL DEF** blocks. After a power interruption the TNC returns to the most recently selected setting.

Inserting the "/" character

- ▶ In the **Programming** mode you select the block in which the character is to be inserted



- ▶ Select the INSERT soft key

Erasing the "/" character

- ▶ In the **Programming** mode you select the block in which the character is to be deleted



- ▶ Select the REMOVE soft key

Test run and program run

17.8 Optional program-run interruption

17.8 Optional program-run interruption

Application

The TNC optionally interrupts program run at blocks containing M1. If you use M1 in the Program Run mode, the TNC does not switch off the spindle or coolant.



- ▶ Do not interrupt program run or test run at blocks containing M1: Set soft key to **OFF**



- ▶ Interrupt program run or test run at blocks containing M1: Set soft key to **ON**

18

MOD functions

MOD functions

18.1 MOD function

18.1 MOD function

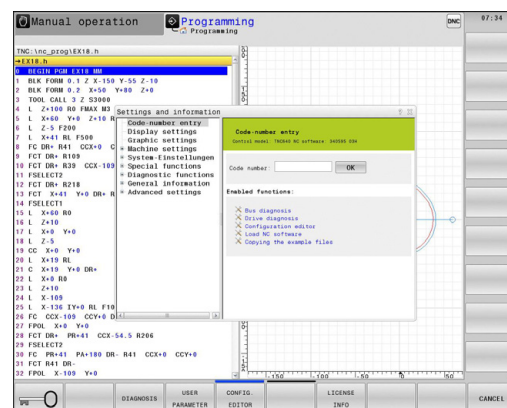
The MOD functions provide additional input possibilities and displays. In addition you can enter code numbers to enable access to protected areas.

Selecting MOD functions

Open the pop-up window with the MOD functions:

MOD

- ▶ To select the MOD functions, Press the **MOD** key. The TNC opens a pop-up window displaying the available MOD functions.



Changing the settings

As well as with the mouse, navigation with the keyboard is also possible in the MOD functions:

- ▶ Switch from the input area in the right window to the MOD function selections in the left window with the tab key
- ▶ Select MOD function
- ▶ Switch to the input field with the tab key or ENT key
- ▶ Enter value according to function and confirm with **OK** or make selection and confirm with **Apply**



If more than one possibility is available for a particular setting, you can superimpose a window listing all of the given possibilities by pressing the GOTO key. Select the setting with the ENT key. If you don't want to change the setting, close the window again with END.

Exiting MOD functions

- ▶ To close the MOD functions, Press the CANCEL or **END** key

Overview of MOD functions

The following functions are available independent of the selected operating mode:

Code-number entry

- Code number

Display settings

- Digital readouts
- Unit of measurement (mm/inches) for position display
- Program entry for MDI
- Show time of day
- Show the info line

Graphic settings

- Model type
- Model quality

Machine settings

- Kinematics
- Traverse limits
- Tool-usage file
- External access

System settings

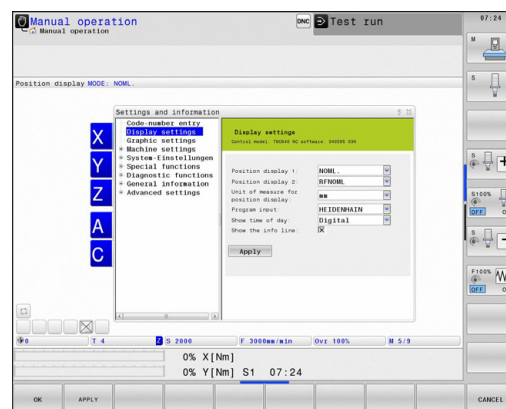
- Set the system time
- Define the network connection
- Network: IP configuration

Diagnostic functions

- Bus diagnosis
- Diagnosis of drives
- HEROS information

General information

- Software version
- FCL information
- License information
- Machine times



MOD functions

18.2 Graphic settings

18.2 Graphic settings




With the MOD function **Graphic settings**, you can select the model type and model quality.

Select the graphic settings:

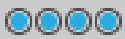



- ▶ In the MOD menu, select the **Graphic settings** group
- ▶ Select the model type
- ▶ Select the model quality
- ▶ Press the **APPLY** soft key
- ▶ Press the **OK** soft key

You have the following simulation parameters for the graphic settings:

Model type

Displayed symbol	Choice	Properties	Application
	3-D	Very true to detail, heavy time and processor consumption	Milling with undercuts, milling-turning operations
	2.5 D	Fast	Milling without undercuts
	No model	Very fast	Line graphics

Model quality

Displayed symbol	Choice	Properties
	Very high	High data transfer rate, exact depiction of tool geometry, depiction of block end points and block numbers possible
	High	High data transfer rate, exact depiction of tool geometry
	Medium	Medium data transfer rate, approximation of tool geometry
	Low	Low data transfer rate, coarse approximation of tool geometry

18.3 Machine settings

External access



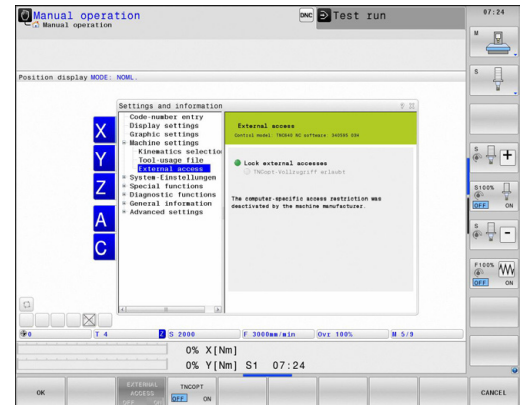
The machine tool builder can configure the external access options. Refer to your machine manual.

Machine-dependent function: With the **TNCOPT** soft key, you can permit or lock access for an external diagnostics or commissioning program.

With the MOD function **External access** you can grant or restrict access to the TNC. If you have restricted the external access it is no longer possible to connect to the TNC and exchange data via a network or a serial connection, e.g. with the TNCremo data transfer software.

Restricting external access:

- ▶ In the MOD menu select the **Machine settings** group
- ▶ Select the **External access** menu
- ▶ Set the **EXTERNAL ACCESS SOFT KEY TO OFF**
- ▶ Press the **OK** soft key



MOD functions

18.3 Machine settings

Computer-specific access control

If your machine tool builder has set up computer-specific access control (machine parameter **CfgAccessCtrl**), you can permit access for up to 32 connections authorized by you. Select **Add new** to create a new connection. The TNC opens an input window for you to enter the connection data.

Access settings

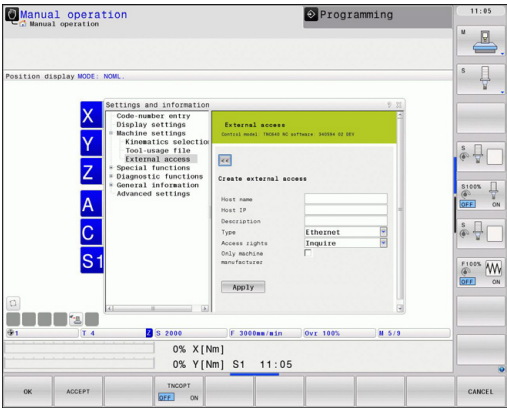
Host name	Host name of the external computer
Host IP	Network address of the external computer
Description	Additional information (text is shown in the overview list)

Type:

Ethernet	Network connection
Com 1	Serial interface 1
COM 2	Serial interface 2

Access rights:

Inquire	The TNC opens a query dialog with external access
Deny	Do not permit network access
Permit	Permit network access without query
Only machine manufacturer	Connection possible only when a code number is entered (machine tool builder)



If you assign the access right **Query** to a connection and access is implemented from this address, the TNC opens a pop-up window. You must permit or deny external access in the pop-up window:

External access	Permission
Yes	Permit once
Always	Permit continuously
Never	Deny continuously
No	Deny once



In the overview list an active connection is shown with a green symbol.
Connections without access rights are shown gray in the overview list.

Entering traverse limits



The **Traverse limits** function must be enabled and adapted by the machine tool builder.
Refer to your machine manual.

The MOD function **Traverse limits** enables you to limit the actually usable tool path within the maximum traverse range. This enables you to define protection zones in each axis to protect a component from collision for example.

To enter traverse limits:

- ▶ In the MOD menu select the **Machine settings** group
- ▶ Select the **Traverse limits** menu
- ▶ Enter the values of the desired axes as a reference value or load the momentary position with the **ACTUAL POSITION CAPTURE** soft key
- ▶ Press the **APPLY** soft key
- ▶ Press the **OK** soft key



The protection zone becomes active automatically as soon as you set a limit in an axis. Settings are kept even after restarting the control.

You can only deactivate the protection zone by deleting all values or pressing the **CLEAR ALL** soft key.

MOD functions

18.3 Machine settings

Tool usage file



The tool usage test function must be enabled by your machine manufacturer. Refer to your machine manual.

With the MOD function **Tool usage file** you can select whether the TNC never, once, or always uses a tool usage file.

To generate a tool usage file:

- ▶ In the MOD menu select the **Machine settings** group
- ▶ Select the **Tool usage file** menu
- ▶ Select the desired setting for the **Program Run, Full Sequence/Single Block** and **Test Run** operating modes
- ▶ Press the **APPLY** soft key
- ▶ Press the **OK** soft key

Select kinematics



The **Select Kinematics** function must be enabled and adapted by the machine manufacturer.
Refer to your machine manual.

You can use this function to test programs whose kinematics does not match the active machine kinematics. If your machine manufacturer saved different kinematic configurations in your machine, you can activate one of these kinematics configurations with the MOD function. When you select a kinematics model for the test run this does not affect machine kinematics.



Danger of collision!

When you switch the kinematics model for machine operation, the TNC implements all of subsequent movements with modified kinematics.

Ensure that you have selected the correct kinematics in the test run for checking your workpiece.

MOD functions

18.4 System settings

18.4 System settings

Set the system time

With the **Set system time** MOD function you can set the time zone, data and time manually or with the aid of an NTP server synchronization.

To set the system time manually:

- ▶ In the MOD menu, select the **System settings** group
- ▶ Press the **SET DATE/TIME** soft key
- ▶ Select your time zone in the **Time zone** area
- ▶ Press the **LOCAL/NTP** soft key in order to select the **Set time manually** entry
- ▶ If required, change the datum and the time
- ▶ Press the **OK** soft key

To set the system time with the aid of an NTP server:

- ▶ In the MOD menu, select the **System settings** group
- ▶ Press the **SET DATE/TIME** soft key
- ▶ Select your time zone in the **Time zone** area
- ▶ Press the **LOCAL/NTP** soft key in order to synchronize the time entry through the NTP server
- ▶ Enter the host name or the URL of an NTP server
- ▶ Press the **ADD** soft key
- ▶ Press the **OK** soft key

18.5 Select the position display

Application

In the **Manual Operation** mode and the **Program Run, Full Sequence** and **Program Run, Single Block** modes of operation, you can select the type of coordinates to be displayed:

The figure at right shows the different tool positions:

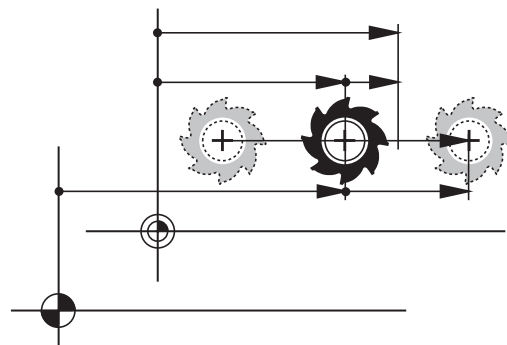
- Initial position
- Target position of the tool
- Workpiece datum
- Machine datum

The TNC position displays can show the following coordinates:

Function	Display
Nominal position: the value presently commanded by the TNC	NOML.
Actual position; current tool position	ACTL.
Reference position; the actual position relative to the machine datum	REF ACTL
Reference position; the nominal position relative to the machine datum	REF NOML
Servo lag; difference between nominal and actual positions (following error)	LAG
Distance remaining to the programmed position in the input system; difference between actual and target positions	ACTDST
Distance remaining to the programmed position with reference to the machine datum; difference between reference and target positions	REFDST
Traverses that were carried out with handwheel superimpositioning (M118)	M118

With the MOD function **Position display 1**, you can select the position display in the status display.

With the MOD function **Position display 2**, you can select the position display in the additional status display.



MOD functions

18.6 Setting the unit of measure

18.6 Setting the unit of measure

Application

This MOD function determines whether the coordinates are displayed in millimeters (metric system) or inches.

- Metric system: e.g. X = 15.789 (mm), the value is displayed to 3 decimal places
- Inch system: e.g. X = 0.6216 (inches), value is displayed to 4 decimal places

If you would like to activate the inch display, the TNC shows the feed rate in inch/min. In an inch program you must enter the feed rate larger by a factor of 10.

18.7 Displaying operating times

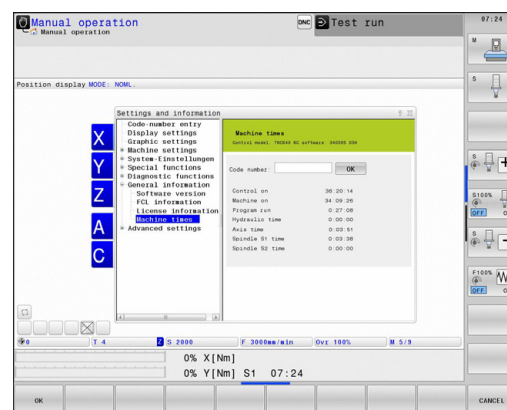
Application

The **MACHINE TIME** MOD function enables you to see various types of operating times:

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



The machine tool builder can provide further operating time displays. Refer to your machine manual.



18.8 Software numbers

Application

The following software numbers are displayed on the TNC screen after the "Software version" MOD function has been selected:

- **Control model:** Designation of the control (managed by HEIDENHAIN)
- **NC SW:** Number of the NC software (managed by HEIDENHAIN)
- **NCK:** Number of the NC software (managed by HEIDENHAIN)
- **PLC SW:** Number and name of the PLC software (managed by your machine tool builder)

In the "FCL information" MOD function, the TNC shows the following information:

- **Development level (FCL=Feature Content Level):**
Development level of the software installed on the control, see "Feature Content Level (upgrade functions)", page 11

18.9 Entering the code number

Application

The TNC requires a code number for the following functions:

Function	Code number
Selecting user parameters	123
Configuring an Ethernet card	NET123
Enabling special functions for Q parameter programming	555343

MOD functions

18.10 Setting up data interfaces

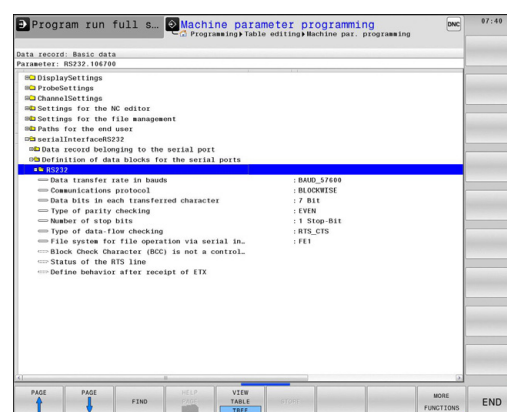
18.10 Setting up data interfaces

Serial interfaces on the TNC 640

The TNC 640 automatically uses the LSV2 transmission protocol for serial data transfer. The LSV2 protocol is permanent and cannot be changed except for setting the baud rate (machine parameter **baudRateLsv2**). You can also specify another type of transmission (interface). The settings described below are therefore effective only for the respective newly defined interface.

Application

To set up a data interface, select the file manager (PGM MGT) and press the MOD key. Press the MOD key again and enter the code number 123. The TNC shows the user parameter **GfgSerialInterface**, in which you can enter the following settings:



Setting the RS-232 interface

Open the RS232 folder. The TNC then displays the following settings:

Setting the BAUD RATE (baudRate)

You can set the BAUD RATE (data transfer speed) from 110 to 115 200 baud.

Setting the protocol (protocol)

The data transfer protocol controls the data flow of a serial transmission (comparable to MP5030 of the iTNC 530).



Here, the BLOCKWISE setting designates a form of data transfer where data is transmitted in blocks. This is not to be confused with the blockwise data reception and simultaneous blockwise processing by older TNC contouring controls. Blockwise reception of an NC program and simultaneous machining of the program is not possible!

Data transmission protocol	Selection
Standard data transmission (transmission line-by-line)	STANDARD
Packet-based data transfer	BLOCKWISE
Transmission without protocol (only character-by-character)	RAW_DATA

Setting data bits (dataBits)

By setting the data bits you define whether a character is transmitted with 7 or 8 data bits.

Check parity (parity)

The parity bit helps the receiver to detect transmission errors. The parity bit can be formed in three different ways:

- No parity (NONE): There is no error detection
- Even parity (EVEN): Here there is an error if the receiver finds that it has received an odd number of set bits
- Odd parity (ODD): Here there is an error if the receiver finds that it has received an even number of set bits

Setting the stop bits (stopBits)

The start bit and one or two stop bits enable the receiver to synchronize to every transmitted character during serial data transmission.

MOD functions

18.10 Setting up data interfaces

Setting handshaking (flowControl)

By handshaking, two devices control data transfer between them. A distinction is made between software handshaking and hardware handshaking.

- No data flow checking (NONE): Handshaking is not active
- Hardware handshaking (RTS_CTS): Transmission stop is active through RTS
- Software handshaking (XON_XOFF): Transmission stop is active through DC3 (XOFF)

File system for file operations (fileSystem)

In **fileSystem** you define the file system for the serial interface. This machine parameter is not required if you don't need a special file system.

- EXT: Minimum file system for printers or non-HEIDENHAIN transmission software. Corresponds to the EXT1 and EXT2 modes of earlier TNC controls.
- FE1: Communication with the TNCserver PC software or an external floppy disk unit.

Block Check Character (bccAvoidCtrlChar)

With Block Check Character (optional) no control character, you determine whether the checksum can correspond to a control character.

- TRUE: The checksum does not correspond to a control character
- FALSE: The checksum can correspond to a control character

Condition of RTS line (rtsLow)

With Condition of RTS line (optional) you determine whether the "low" level is active in idle state.

- TRUE: Level is "low" in idle state
- FALSE: Level is not "low" in idle state

Define behavior after reception of ETX (noEotAfterEtx)

With define behavior after reception of ETX (optional) you determine whether the EOT character is sent after the ETX character was received.


- TRUE: The EOT character is not sent
- FALSE: The EOT character is sent

Settings for data transfer with the TNCserver PC software




Enter the following settings in the user parameters
(**serialInterfaceRS232 / definition of data blocks for the serial ports / RS232**):

Parameters	Selection
Data transfer rate in baud	Has to match the setting in TNCserver
Data transmission protocol	BLOCKWISE
Data bits in each transferred character	7 bits
Type of parity checking	EVEN
Number of stop bits	1 stop bit
Specify type of handshake:	RTS_CTS
File system for file operations	FE1

Setting the operating mode of the external device
(fileSystem)



The functions "Transfer all files," "Transfer selected file," and "Transfer directory" are not available in the FE2 and FEX modes.

Icon	External device	Operating mode
	PC with HEIDENHAIN TNCremo data transfer software	LSV2
	HEIDENHAIN floppy disk units	FE1
	Non-HEIDENHAIN devices such as printers, scanners, punchers, PC without TNCremo	FEX

Data transfer software

For transfer of files to and from the TNC, we recommend using the HEIDENHAIN TNCremo data transfer software. With TNCremo, data transfer is possible with all HEIDENHAIN controls via the serial interface or the Ethernet interface.



You can download the current version of TNCremo free of charge from the HEIDENHAIN Filebase (www.heidenhain.de, <Documentation and Information>, <Software>, <Download area>, <PC Software>, <TNCremo>).

System requirements for TNCremo:

- PC with 486 processor or higher
- Windows XP, Windows Vista, Windows 7, Windows 8 operating system
- 16 MB RAM
- 5 MB free memory space on your hard disk
- An available serial interface or connection to the TCP/IP network

Installation under Windows

- ▶ Start the SETUP.EXE installation program with the file manager (Explorer)
- ▶ Follow the setup program instructions

Starting TNCremo under Windows

- ▶ Click on <Start>, <Programs>, <HEIDENHAIN Applications>, <TNCremo>

When you start TNCremo for the first time, TNCremo automatically tries to set up a connection with the TNC.

18.10 Setting up data interfaces

Data transfer between the TNC and TNCremo



Before you transfer a program from the TNC to the PC, you must make absolutely sure that you have already saved the program currently selected on the TNC. The TNC saves changes automatically when you switch the mode of operation on the TNC, or when you select the file manager via the PGM MGT key.

Check whether the TNC is connected to the correct serial port on your PC or to the network.

Once you have started TNCremo, you will see a list of all files that are stored in the active directory in the upper section of the main window **1**. Using <File>, <Change directory>, you can select any drive or another directory on your PC.

If you want to control data transfer from the PC, establish the connection with your PC in the following manner:

- ▶ Select <File>, <Setup connection>. TNCremo now receives the file and directory structure from the TNC and displays this at the bottom left of the main window **2**
- ▶ To transfer a file from the TNC to the PC, select the file in the TNC window with a mouse click and drag and drop the highlighted file into the PC window **1**
- ▶ To transfer a file from the PC to the TNC, select the file in the PC window with a mouse click and drag and drop the highlighted file into the TNC window **2**

If you want to control data transfer from the TNC, establish the connection with your PC in the following manner:

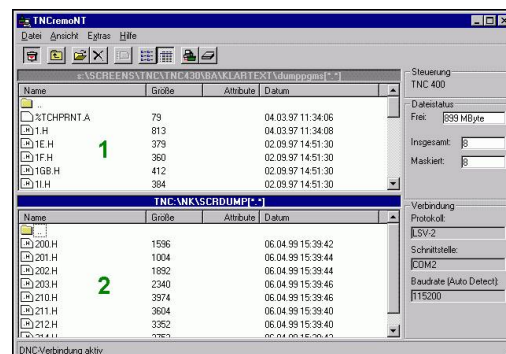
- ▶ Select <Extras>, <TNCserver>. TNCremo is now in server mode. It can receive data from the TNC and send data to the TNC
- ▶ You can now call the file management functions on the TNC by pressing the key **PGM MGT** see "Data transfer to/from an external data medium", page 136, in order to transfer the desired files

End TNCremo

Select <File>, <Exit>



Refer also to the TNCremo context-sensitive help texts where all of the functions are explained in more detail. The help texts must be called with the F1 key.



18.11 Ethernet interface

Introduction

The TNC is shipped with a standard Ethernet card to connect the control as a client in your network. The TNC transmits data via the Ethernet card with

- the **smb** protocol (Server Message Block) for Windows operating systems, or
- the **TCP/IP** protocol family (Transmission Control Protocol/Internet Protocol) and with support from the NFS (Network File System)

Connection options

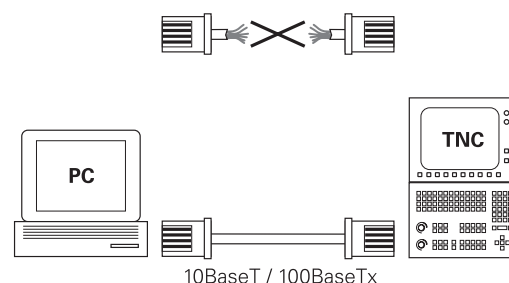
You can connect the Ethernet card in your TNC to your network through the RJ45 connection (X26, 100BaseTX or 10BaseT), or directly to a PC. The connection is metalically isolated from the control electronics.

For a 100BaseTX or 10BaseT connection you need a Twisted Pair cable to connect the TNC to your network.



The maximum cable length between TNC and a node depends on the quality grade of the cable, the sheathing and the type of network (100BaseTX or 10BaseT).

No great effort is required to connect the TNC directly to a PC that has an Ethernet card. Simply connect the TNC (port X26) and the PC with an Ethernet crossover cable (trade names: crossed patch cable or STP cable).



Configuring the TNC



Make sure that the person configuring your TNC is a network specialist.

- ▶ Press the MOD key in the **Programming** operating mode and enter the code number NET123
- ▶ In the file manager, press the **NETWORK** soft key **NET**

18.11 Ethernet interface

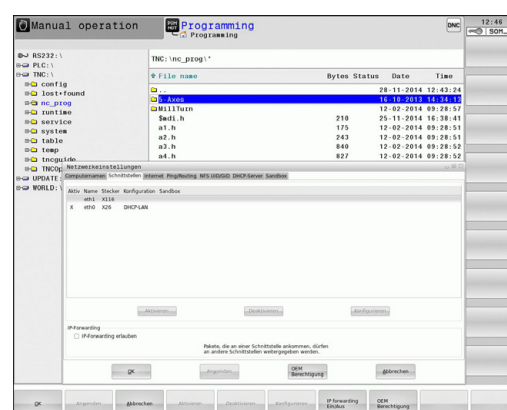
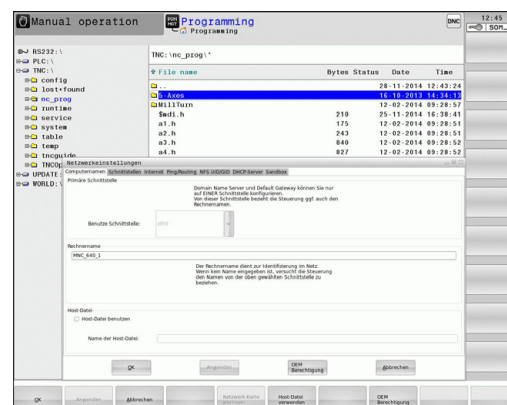
General network settings

- Press the **CONFIGURE NETWORK** soft key to enter the general network settings. The **Computer name** tab is active:

Setting	Meaning
Primary interface	Name of the Ethernet interface to be integrated in your company network. Only active if a second, optional Ethernet interface is available on the control hardware
Computer name	Name displayed for the TNC in your company network
Host file	Only required for special applications: Name of a file in which the assignments of IP addresses to computer names is defined

- Select the **Interfaces** tab to enter the interface settings:

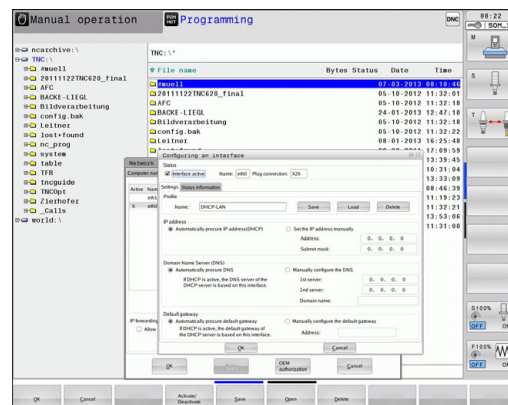
Setting	Meaning
Interface list	List of the active Ethernet interfaces. Select one of the listed interfaces (via mouse or arrow keys) <ul style="list-style-type: none"> ■ Activate button: Activate the selected interface (an X appears in the Active column) ■ Deactivate button: Deactivate the selected interface (- in the Active column) ■ Configuration button: Open the configuration menu
Allow IP forwarding	This function must be kept deactivated. Only activate this function if external access via the second, optional Ethernet interface of the TNC is necessary for diagnostic purposes. Only do so after instruction by our Service Department



- Press the **Configuration** button to open the Configuration menu:

Setting	Meaning
Status	<ul style="list-style-type: none"> ■ Active interface: Connection status of the selected Ethernet interface ■ Name: Name of the interface you are currently configuring ■ Plug connection: Number of the plug connection of this interface on the logic unit of the control
Profile	<p>Here you can create or select a profile in which all settings shown in this window are stored. HEIDENHAIN provides two standard profiles:</p> <ul style="list-style-type: none"> ■ DHCP-LAN: Settings for the standard TNC Ethernet interface, should work in a standard company network ■ MachineNet: Settings for the second, optional Ethernet interface; for configuration of the machine network <p>Press the corresponding buttons to save, load and delete profiles</p>
IP address	<ul style="list-style-type: none"> ■ Option Automatically procure IP address: The TNC is to procure the IP address from the DHCP server ■ Option Manually set IP address: Manually define the IP address and subnet mask. Input: Four numerical values separated by points, in each field, e.g. 160.1.180.20 and 255.255.0.0
Domain Name Server (DNS)	<ul style="list-style-type: none"> ■ Option Automatically procure DNS: The TNC is to automatically procure the IP address of the domain name server ■ Option Manually configure DNS: Manually enter the IP addresses of the servers and the domain name
Default gateway	<ul style="list-style-type: none"> ■ Option Automatically procure default GW: The TNC is to automatically procure the default gateway ■ Option Manually configure default GW: Manually enter the IP addresses of the default gateway

- Apply the changes with the **OK** button, or discard them with the **Cancel** button



18.11 Ethernet interface

- ▶ Select the **Internet** tab.

Setting	Meaning
Proxy	<ul style="list-style-type: none"> ■ Direct connection to Internet /NAT: The control forwards Internet inquiries to the default gateway and from there they must be forwarded through network address translation (e.g.. if a direct connection to a modem is available) ■ Use proxy: Define the Address and Port of the Internet router in your network, ask your network administrator for the correct address and port

Telemaintenance The machine manufacturer configures the server for telemaintenance here. Changes must always be made in agreement with your machine tool builder

- ▶ Select the **Ping/Routing** tab to enter the ping and routing settings:

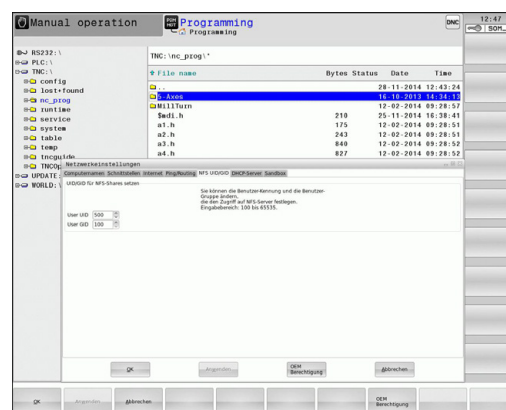
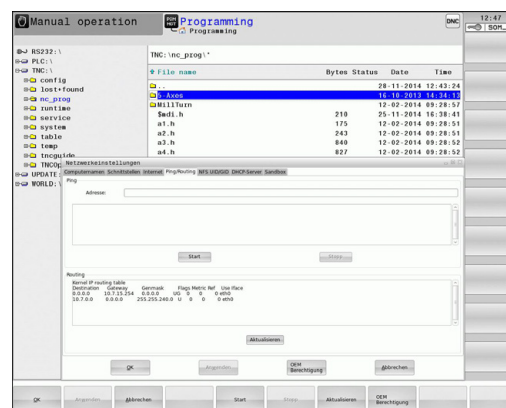
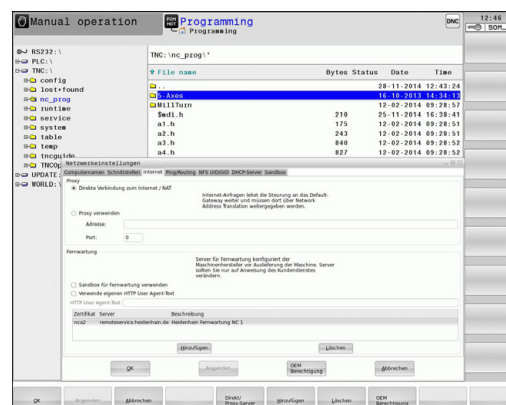
Setting	Meaning
Ping	<p>In the Address: field, enter the IP number for which you want to check the network connection. Input: four numerical values separated by periods, e.g. 160.1.180.20. As an alternative, you can enter the name of the computer whose connection you want to check</p> <ul style="list-style-type: none"> ■ Press the Start button to begin the test. The TNC shows the status information in the Ping field ■ Press the Stop button to conclude the test

Routing For network specialists: Status information of the operating system for the current routing

- Press the **Update** button to refresh the routing information

- ▶ Select the **NFS UID/GID** tab to enter the user and group identifications:

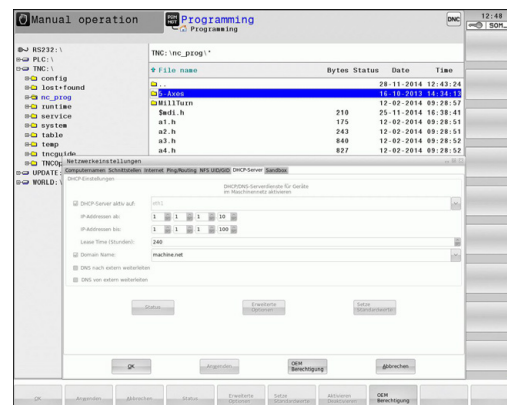
Setting	Meaning
Set UID/GID for NFS shares	<ul style="list-style-type: none"> ■ User ID: Definition of which user identification the end user uses to access files in the network. Ask your network specialist for the proper value ■ Group ID: Definition of the group identification with which you access files in the network. Ask your network specialist for the proper value



► **DHCP server:** Settings for automatic network configuration

Setting	Meaning
DHCP server	<ul style="list-style-type: none"> ■ IP addresses from: Define the IP address as of which the TNC is to derive the pool of dynamic IP addresses. The TNC transfers the values that appear dimmed from the static IP address of the defined Ethernet interface; these values cannot be edited. ■ IP addresses to: Define the IP address up to which the TNC is to derive the pool of dynamic IP addresses ■ Lease Time (hours): Time within which the dynamic IP address is to remain reserved for a client. If a client logs on within this time, the TNC reassigns the same dynamic IP address. ■ Domain name: Here you can define a name for the machine network if required. This is necessary if the same names are assigned in the machine network and in the external network, for example. ■ Forward DNS externally: If IP Forwarding is active (Interfaces tab) and the option is active, you can specify that the name resolution for devices in the machine network can also be used by the external network. ■ Forward DNS from outside: If IP Forwarding is active (Interfaces tab) and the option is active, you can specify that the TNC is to forward DNS inquiries from devices within the machine network to the name server of the external network if the DNS server of the MC cannot answer the inquiry. ■ Status button: Call an overview of the devices that are provided with a dynamic IP address in the machine network. You can also select settings for these devices. ■ Additional options button: Additional settings for the DNS/DHCP server. ■ Set standard values button: Set factory settings.

► **Sandbox:** Changes must always be made in agreement with your machine tool builder

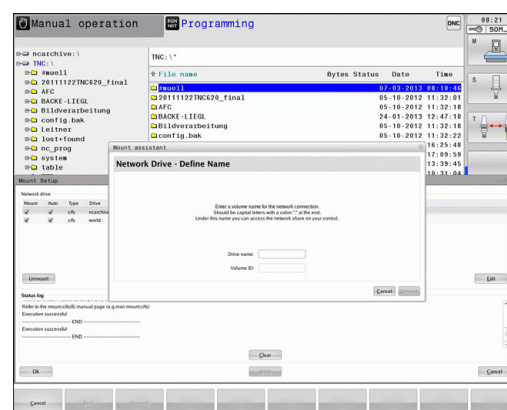
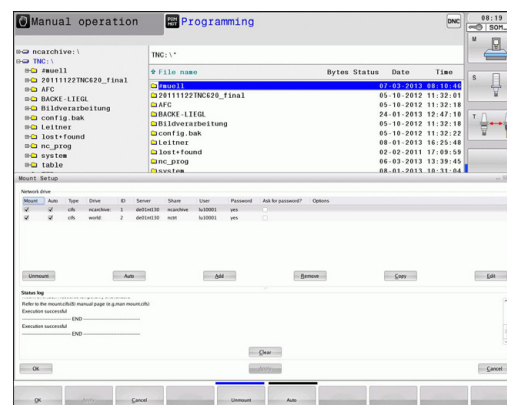


18.11 Ethernet interface

Network settings specific to the device

- Press the **DEFINE NETWORK CONNECTN.** soft key to enter the network settings for a specific device. You can define any number of network settings, but you can manage only seven at one time

Setting	Meaning
Network drive	<p>List of all connected network drives. The TNC shows the respective status of the network connections in the columns:</p> <ul style="list-style-type: none"> ■ Mount: Network drive connected / not connected ■ Auto: Network drive is to be connected automatically/manually ■ Type: Type of network connection. cifs and nfs are possible ■ Drive: Designation of the drive on the TNC ■ ID: Internal ID that identifies if a mount point has been used for more than one connection ■ Server: Name of the server ■ Authorization name: Name of the directory on the server that the TNC is to access ■ User: User name with which the user logs on to the network ■ Password: Network drive password protected / not protected ■ Request password?: Request / Do not request password during connection ■ Options: Display additional connection options <p>To manage the network drives, use the screen buttons.</p> <p>To add network drives, use the Add button: The TNC then starts the connection wizard, which guides you by dialog through the required definitions.</p>
Status log	<p>Display of status information and error messages.</p> <p>Press the Clear button to delete the contents of the Status Log window.</p>






18.12 Firewall

Application

You can set up a firewall for the primary network interface of the control. It can be configured so that incoming network traffic is blocked and/or a message is displayed depending on the sender and the service. However, the firewall cannot be started for the second network interface of the control if it is active as DHCP server.

Once the firewall has been activated, a symbol appears at the lower right in the taskbar. The symbol changes depending on the safety level that the firewall was activated with, and informs about the level of the safety settings:

Icon	Meaning
	No firewall protection provided although it was activated in the configuration. This can happen, for example, if PC names were used in the configuration for which there are no equivalent IP addresses as yet.
	Firewall active with medium safety level.
	Firewall active with high safety level. (All services except for the SSH are blocked)



Have the standard settings checked by your network specialist and change them if necessary.

The settings in the additional tab **SSH settings** are in preparation for future enhancements and currently have no function.

Configuring the firewall

Make your firewall settings as follows:

- ▶ Use the mouse to open the task bar at the bottom edge of the screen(see "Window Manager", page 86)
- ▶ Press the green HEIDENHAIN button to open the JH menu.
- ▶ Select the **Settings** menu item
- ▶ Select the **Firewall** menu item

HEIDENHAIN recommends activating the firewall with the prepared default settings:

- ▶ Set the **Active** option to enable the firewall
- ▶ Press the **Set standard values** button to activate the default settings recommended by HEIDENHAIN.
- ▶ Close the dialog with **OK**

18.12 Firewall

Firewall settings

Option	Meaning
Active	Switching the firewall on or off
Interface:	Selection of the eth0 interface usually corresponds to X26 of the MC main computer. eth1 corresponds to X116. You can check this in the network settings in the Interfaces tab. On main computer units with two Ethernet interfaces, the DHCP server is active by default for the second (non-primary) interface for the machine network. With this setting it is not possible to activate the firewall for eth1 because the firewall and the DHCP server exclude themselves mutually
Report other inhibited packets:	Firewall active with high safety level. (All services except for the SSH are blocked)
Inhibit ICMP echo answer:	If this option is set, the control no longer answers to a PING request.
Service	<p>This column contains the short names of the services that are configured with this dialog. For the configuration it is not important here whether the services themselves have been started</p> <ul style="list-style-type: none"> ■ LSV2 contains the functionality for TNCRemoNT and Teleservice, as well as the HEIDENHAIN DNC interface (ports 19000 to 19010) ■ SMB only refers to incoming SMB connections, i.e. if a Windows release is made on the NC. Outgoing SMB connections (i.e. if a Windows release is connected to the NC) cannot be prevented. ■ SSH stands for the Secure Shell protocol (port 22). As of HEROS 504, the LSV2 can be executed safely tunneled via this SSH protocol. ■ VNC protocol means access to the screen contents. If this service is blocked, the screen content can no longer be accessed, not even with the Teleservice programs from HEIDENHAIN (e.g. screenshot). If this service is blocked, the VNC configuration dialog shows a warning from HEROS that VNC is disabled in the firewall.

Option	Meaning
Method	Under Method you can configure whether the service should not be available to anyone (Prohibit all), available to everyone (Permit all) or only available to some (Permit some). If you set Permit some you must also specify the computer (under Computer) that you wish to grant access to the respective service. If you do not specify any computer under Computer , the setting Prohibit all will become active automatically when the configuration is saved.
Log	If Log is activated, a "red" message is output if a network package for this service was blocked. A "blue" message is output if a network package for this service was accepted.
Computer	If the setting Permit some is selected under Method , the relevant computers can be specified here. The computers can be entered with their IP addresses or host names separated by commas. If a host name is used, the system checks upon closing or saving of the dialog whether the host name can be translated into an IP address. If this is not the case, the user receives an error message and the dialog box is not closed. If you enter a valid host name, this host name will be translated into an IP address upon every startup of the control. If a computer that was entered with its name changes its IP address, you may have to restart the control or formally change the firewall configuration to ensure that the control uses the new IP address for a host name in the firewall.
Advanced options	These settings are only intended for your network specialists.
Set standard values	Resets the settings to the default values recommended by HEIDENHAIN

MOD functions

18.13 Configure HR 550 FS wireless handwheel

18.13 Configure HR 550 FS wireless handwheel

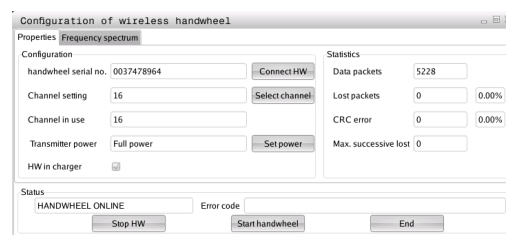
Application

Press the **SET UP WIRELESS HANDWHEEL** soft key to configure the HR 550 FS wireless handwheel. The following functions are available:

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

Assigning the handwheel to a specific handwheel holder

- ▶ Make sure that the handwheel holder is connected to the control hardware.
- ▶ Place the wireless handwheel you want to assign to the handwheel holder in the handwheel holder
- ▶ Press the **MOD** key to select the MOD function
- ▶ Select the **Machine settings** menu
- ▶ Select the configuration menu for the wireless handwheel:
Press the **SET UP WIRELESS HANDWHEEL** soft key
- ▶ Click the **Connect HR** button: The TNC saves the serial number of the wireless handwheel located in the handwheel holder and shows it in the configuration window to the left of the **Connect HR** button
- ▶ To save the configuration and exit the configuration menu, press the **END** button

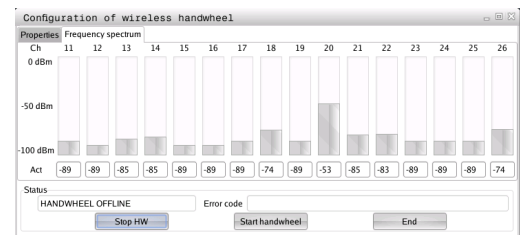
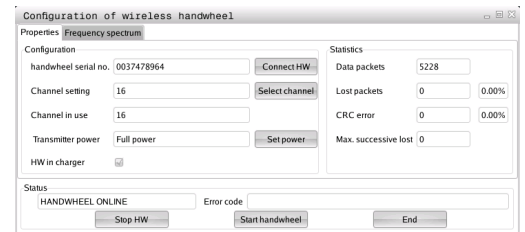


Configure HR 550 FS wireless handwheel 18.13

Setting the transmission channel

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

- ▶ Press the **MOD** key to select the MOD function
- ▶ Select the **Machine settings** menu
- ▶ Select the configuration menu for the wireless handwheel:
Press the **SET UP WIRELESS HANDWHEEL** soft key
- ▶ Click the **Frequency spectrum** tab
- ▶ Click the **Stop HR** button: The TNC stops the connection to the wireless handwheel and determines the current frequency spectrum for all of the 16 available channels
- ▶ Memorize the number of the channel with the least amount of radio traffic (smallest bar)
- ▶ Click the **Start handwheel** button to reactivate the wireless handwheel
- ▶ Click the **Properties** tab
- ▶ Click the **Select channel** button: The TNC shows all available channel numbers. Click the channel number for which the TNC determined the least amount of radio traffic
- ▶ To save the configuration and exit the configuration menu, press the **END** button

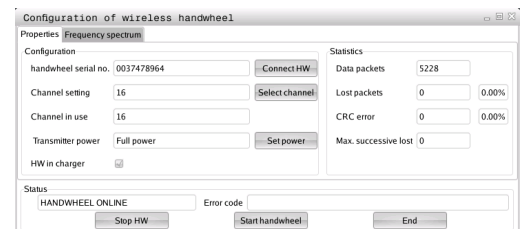


Selecting the transmitter power



Please keep in mind that the transmission range of the wireless handwheel decreases when the transmitter power is reduced.

- ▶ Press the **MOD** key to select the MOD function
- ▶ Select the **Machine settings** menu
- ▶ Select the configuration menu for the wireless handwheel:
Press the **SET UP WIRELESS HANDWHEEL** soft key
- ▶ Click the **Set power** button: The TNC shows the three available power settings. Click the desired setting
- ▶ To save the configuration and exit the configuration menu, press the **END** button



18.13 Configure HR 550 FS wireless handwheel

Statistical data

To display the statistical data, proceed as follows:

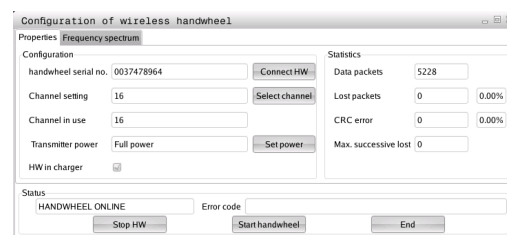
- ▶ Press the **MOD** key to select the MOD function
- ▶ Select the **Machine settings** menu
- ▶ To select the configuration menu for the wireless handwheel, press the **SET UP WIRELESS HANDWHEEL** soft key: The TNC displays the configuration menu with the statistical data

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

If the reception quality is poor so that a proper and safe stop of the axes cannot be ensured anymore, an emergency-stop reaction of the wireless handwheel is triggered.

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

If this occurs, try to improve the transmission quality by selecting another channel (see "Setting the transmission channel", page 641) or by increasing the transmitter power (see "Selecting the transmitter power", page 641).



18.14 Load machine configuration

Application



Caution: Data loss!

The TNC overwrites your machine configuration when you load (restore) a backup. The overwritten machine data will be lost in the process. You can no longer undo this process!

Your machine tool builder can provide you a backup with a machine configuration. After entering the keyword **RESTORE**, you can load the backup on your machine or programming station. Proceed as follows to load the backup:

- ▶ In the MOD dialog, enter the keyword **RESTORE**
- ▶ In the TNC's file manager, select the backup file (e.g. BKUP-2013-12-12_.zip). The TNC opens a pop-up window for the backup
- ▶ Press the emergency stop
- ▶ Press the **OK** soft key to start the backup process

19

**Tables and
overviews**

Tables and overviews

19.1 Machine-specific user parameters

19.1 Machine-specific user parameters

Application

The parameter values are entered in the **configuration editor**.



To enable you to set machine-specific functions for users, your machine tool builder can define which machine parameters are available as user parameters. Furthermore, your machine tool builder can integrate additional machine parameters, which are not described in the following, into the TNC. Refer to your machine manual.

The machine parameters are grouped as parameter objects in a tree structure in the configuration editor. Each parameter object has a name (e.g. **Settings for screen displays**) that gives information about the parameters it contains. A parameter object (entity) is marked with an "E" in the folder symbol in the tree structure. Some machine parameters have a key name to identify them unambiguously. The key name assigns the parameter to a group (e.g. X for X axis). The respective group folder bears the key name and is marked by a "K" in the folder symbol.



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. To display the actual system names of the parameters, press the key for the screen layout key and then the **SHOW SYSTEM NAME** soft key. Follow the same procedure to return to the standard display.








Parameters not yet active and objects appear dimmed. These can be activated with the **MORE FUNCTIONS** and **INSERT** soft key.

The TNC saves a modification list of the last 20 changes to the configuration data. To restore modifications, select the corresponding line and press the **MORE FUNCTIONS** and **DISCARD CHANGES** soft key.




Calling the configuration editor and changing parameters

- ▶ Select the **Programming** mode of operation
- ▶ Press the **MOD** key
- ▶ Enter the code number **123**
- ▶ Changing parameters
- ▶ Press the **END** soft key to exit the configuration editor
- ▶ Press the **SAVE** soft key to save changes

The icon at the beginning of each line in the parameter tree shows additional information about this line. The icons have the following meanings:

-  Branch exists but is closed
-  Branch is open
-  Empty object, cannot be opened
-  Initialized machine parameter
-  Uninitialized (optional) machine parameter
-  Can be read but not edited
-  Can neither be read nor edited

The type of the configuration object is identified by its folder symbol:

-  Key (group name)
-  List
-  Entity (parameter object)

Displaying help texts

The **HELP** key enables you to call a help text for each parameter object or attribute.

If the help text does not fit on one page (1/2 is then displayed at the upper right, for example), press the **HELP PAGE** soft key to scroll to the second page.

To exit the help text, press the **HELP** key again.

Additional information, such as the unit of measure, the initial value, or a selection list, is also displayed. If the selected machine parameter matches a parameter in the previous control model, the corresponding MP number is shown.

Tables and overviews

19.1 Machine-specific user parameters

Parameter list

Parameter settings

DisplaySettings

Settings for screen display

Sequence of displayed axes

[0] to [7]

Depends on available axes

Type of position display in position window

NOMINAL

ACTUAL

REF ACTL

REF NOML

LAG

ACTUAL DIST

DIST

M 118

Type of position display in status display

NOMINAL

ACTUAL

REF ACTL

REF NOML

LAG

ACTUAL DIST

DIST

M 118

Definition of decimal separation characters for position display

.

Display of feed rate in Manual Operation mode

at axis key: Only show feed rate when axis-direction key is pressed

always minimum: Always show feed rate

Display of spindle position in the position display

during closed loop: Only show spindle position when spindle is in position control

during closed loop and M5: Show spindle position when spindle is in position control and with M5

Show or hide Preset table soft key

True: Do not display Preset table soft key

False: Display Preset table soft key

Parameter settings

DisplaySettings

Display step for individual axes

List of all available axes

Display step for position display in mm or degrees

0.1

0.05

0.01

0.005

0.001

0.0005

0.0001

0.00005 (Option #23)

0.00001 (Option #23)

Display step for position display in inches

0.005

0.001

0.0005

0.0001

0.00005 (Option #23)

0.00001 (Option #23)

DisplaySettings

Definition of unit of measure valid for the display

metric: Use metric system

inch: Use inch system

DisplaySettings

Format of NC programs and display of cycles

Program input in HEIDENHAIN conversational text or in DIN/ISO

HEIDENHAIN: Program input in BA MDI in conversational text dialog

ISO: Program input in Positioning with MDI mode of operation in DIN/ISO

19.1 Machine-specific user parameters

Parameter settings

DisplaySettings

Setting the NC and PLC dialog language

NC dialog language

ENGLISH

GERMAN

CZECH

FRENCH

ITALIAN

SPANISH

PORTUGUESE

SWEDISH

DANISH

FINNISH

DUTCH

POLISH

HUNGARIAN

RUSSIAN

CHINESE

CHINESE_TRAD

SLOVENIAN

KOREAN

NORWEGIAN

ROMANIAN

SLOVAK

TURKISH

PLC dialog language

See NC dialog language

PLC error message language

See NC dialog language

Help language

See NC dialog language

Parameter settings

DisplaySettings

Behavior with control start-up

Acknowledge "Power interrupted" message

TRUE: Control start-up is not continued until the message has been acknowledged

FALSE: "Power interrupted" message not displayed

DisplaySettings

Display mode for time display

Selection for display mode in the time display

Analog

Digital

Logo

Analog and Logo

Digital and Logo

Analog on Logo

Digital on Logo

DisplaySettings

Link row On/Off

Display setting for link row

OFF: Deactivate the information line in the operating mode line

ON: Activate the information line in the operating mode line

DisplaySettings

Settings for 3-D graphic simulation

Model type of 3-D graphic simulation

3-D (processor-intensive): Model display for complex machining with undercuts

2.5-D: Model display for 3-axis machining

No Model: Model display deactivated

Model quality of 3-D graphic simulation

very high: High resolution; Display of block end points possible

high: High resolution

medium: Medium resolution

low: Low resolution

DisplaySettings

Settings for position display

Position display with TOOL CALL DL

As Tool Length: The programmed oversize DL is considered as a tool length modification for display of the workpiece-oriented position

As Workpiece Oversize: The programmed oversize DL is considered as a workpiece oversize for display of the workpiece-oriented position

19.1 Machine-specific user parameters

Parameter settings

ProbeSettings

Configuration of tool measurement

TT140_1

M function for spindle orientation

-1: Spindle orientation directly via NC**0: Function inactive****1 to 999: Number of M function for spindle orientation**

Probing routine

MultiDirections: Probe from several directions**SingleDirection: Probe from one direction**

Probing direction for tool radius measurement

X_Positive, Y_Positive, X_Negative, Y_Negative, Z_Positive, Z_Negative (depending on tool axis)

Distance of tool lower edge to probe contact upper edge

0.001 to 99.9999 [mm]: Offset of stylus to tool

Rapid traverse in probing cycle

10 to 300 000 [mm/min]: Rapid traverse in probing cycle

Probing feed rate with tool measurement

1 to 3 000 [mm/min]: Probing feed rate with tool measurement

Calculation of probing feed rate

ConstantTolerance: Calculation of probing feed rate with constant tolerance**VariableTolerance: Calculation of probing feed rate with variable tolerance****ConstantFeed: Constant probing feed rate**

Type of speed determination

Automatic: Determine speed automatically**MinSpindleSpeed: Use minimum spindle speed**

Max. permissible rotational speed on cutting edge

1 to 129 [m/min]: Permissible rotational speed on cutter circumference

Maximum permissible speed with tool measurement

0 bis 1 000 [1/min]: Maximum permissible speed

Maximum permissible measurement error with tool measurement

0.001 to 0.999 [mm]: First maximum permissible measurement error

Maximum permissible measurement error with tool measurement

0.001 bis 0.999 [mm]: Second maximum permissible measurement error

NC stop during tool check

True: When breakage tolerance is exceeded the NC program is stopped**False: The NC program is not stopped**

Parameter settings

NC stop during tool measurement

True: When breakage tolerance is exceeded the NC program is stopped

False: The NC program is not stopped

Modification of tool table during tool check and measurement

AdaptOnMeasure: Table modified after tool measurement

AdaptOnBoth: Table modified after tool check and measurement

AdaptNever: Table not modified after tool check and measurement

Configuring a round stylus

TT140_1

Coordinates of stylus center

[0]: X coordinates of stylus center referenced to machine datum

[1]: Y coordinates of stylus center referenced to machine datum

[2]: Z coordinates of stylus center referenced to machine datum

Safety clearance above stylus for pre-position

0.001 to 99 999.9999 [mm]: Safety clearance in tool axis direction

Safety zone around stylus for pre-position

0.001 to 99 999.9999 [mm]: Safety clearance in the plane vertically to the tool axis

19.1 Machine-specific user parameters

Parameter settings

ChannelSettings

CH_NC

Active kinematics

Kinematics to be activated

List of machine kinematics

Kinematics to be activated with control start-up

List of machine kinematics

Determining the behavior of the NC program

Resetting the machining time with program start

True: Machining time is reset**False: Machining time is not reset**

PLC signal for number of pending machining cycle

Dependent on machine builder

Geometry tolerances

Permissible deviation of circle radius

0.0001 to 0.016 [mm]: Permissible deviation of circle radius on the circle end point compared to circle start point

Configuration of machining cycles

Overlap factor for pocket milling

0.001 to 1.414: Overlap factor for Cycle 4 POCKET MILLING and Cycle 5 CIRCULAR POCKET

Behavior after machining a contour pocket

PosBeforeMachining: Position as before machining a cycle**ToolAxClearanceHeight: Position tool axis to clearance height**

Display "Spindle ?" error message if M3/M4 is not active

on: Output error message**off: Do not output error message**

Display "Enter negative depth" error message

on: Output error message**off: Do not output error message**

Approach behavior on a slot wall in a cylindrical surface

LineNormal: Approach with straight line**CircleTangential: Approach with an arc movement**

M function for spindle orientation in machining cycles

-1: Spindle orientation directly via NC**0: Function inactive****1 to 999: Number of M function for spindle orientation**

Do not display "Plunging type not possible" error message

on: Error message is not displayed

Parameter settings

off: Error message is displayed

Geometry filter for filtering out linear elements

Type of stretch filter

- **Off: No filter active**
- **ShortCut: Leave out single points on polygon**
- **Average: The geometry filter smooths corners**

Maximum distance of filtered to unfiltered contour

0 to 10 [mm]: The filtered out points lie within this tolerance to the resultant distance

Maximum length of distance resulting from filtering

0 to 1000 [mm]: Length over which geometry filtering is effective

19.1 Machine-specific user parameters

Parameter settings

Settings for the editor

Creating backup files

TRUE: Create backup file after editing NC programs

FALSE: Create no backup file after editing NC programs

Cursor behavior after deleting lines

TRUE: Cursor is on previous line after deletion (iTNC behavior)

FALSE: Cursor is on subsequent line after deletion

Cursor behavior with the first and last line

TRUE: All-round cursors permitted at PGM beginning/end

FALSE: All-round cursors not permitted at PGM beginning/end

Line break with multi-line blocks

ALL: Always show lines completely

ACT: Only show lines of the active block completely

NO: Only show lines completely if the block is edited

Activate help graphics with cycle input

TRUE: Fundamentally always show help graphics during input

FALSE: Only show help graphics if the CYCLE HELP soft key is set to ON. The CYCLE HELP OFF/ON soft key is displayed in the Programming mode of operation after pressing the "Screen layout" button

Behavior of soft key row following a cycle input

TRUE: Leave cycle soft key row active after a cycle definition

FALSE: Hide cycle soft key row after a cycle definition

Confirmation request before block is deleted

TRUE: Display confirmation request before deleting an NC block

FALSE: Do not display confirmation request before deleting an NC block

Line number up to which NC program is tested

100 to 100000: Program length for which geometry should be tested

DIN/ISO programming: Block number increment

0 to 250: Increment for generating DIN/ISO blocks in the program

Define programmable axes

TRUE: Use defined axis configuration

FALSE: Use default axis configuration XYZABCUVW

Behavior with paraxial positioning blocks

TRUE: Paraxial positioning blocks permitted

FALSE: Paraxial positioning blocks locked

Line number up to which identical syntax elements are searched for

500 to 400000: Search for selected elements with up/down arrow keys

Parameter settings

Settings for the file manager

Display of dependent files

MANUAL: Dependent files are displayed

AUTOMATIC: Dependent files are not displayed

Path specifications for end users

List with drives and/or directories

Drives and directories entered here are shown by the TNC in the file manager

FN 16 output path for execution

Path for FN 16 output if no path has been defined in the program

FN 16 output path for Programming and Test Run operating modes

Path for FN 16 output if no path has been defined in the program


Serial Interface RS232: see "Setting up data interfaces", page 624

Tables and overviews

19.2 Connector pin layout and connection cables for data interfaces

19.2 Connector pin layout and connection cables for data interfaces

RS-232-C/V.24 interface for HEIDENHAIN devices



The interface complies with the requirements of EN 50 178 for **low voltage electrical separation**.

When using the 25-pin adapter block:

TNC		Conn. cable 365725-xx			Adapter block 310085-01		Conn. cable 274545-xx		
Male	Assignment	Female	Color	Female	Male	Female	Male	Color	Female
1	Do not assign	1		1	1	1	1	White/ Brown	1
2	RXD	2	Yellow	3	3	3	3	Yellow	2
3	TXD	3	Green	2	2	2	2	Green	3
4	DTR	4	Brown	20	20	20	20	Brown	8
5	Signal GND	5	Red	7	7	7	7	Red	7
6	DSR	6	Blue	6	6	6	6		6
7	RTS	7	Gray	4	4	4	4	Gray	5
8	CTR	8	Pink	5	5	5	5	Pink	4
9	Do not assign	9					8	Violet	20
Hsg.	External shield	Hsg.	External shield	Hsg.	Hsg.	Hsg.	Hsg.	External shield	Hsg.

Connector pin layout and connection cables for data interfaces 19.2

When using the 9-pin adapter block:

TNC		Conn. cable 355484-xx		Adapter block 363987-02		Conn. cable 366964-xx			
Male	Assignment	Female	Color	Male	Female	Male	Female	Color	Female
1	Do not assign	1	Red	1	1	1	1	Red	1
2	RXD	2	Yellow	2	2	2	2	Yellow	3
3	TXD	3	White	3	3	3	3	White	2
4	DTR	4	Brown	4	4	4	4	Brown	6
5	Signal GND	5	Black	5	5	5	5	Black	5
6	DSR	6	Violet	6	6	6	6	Violet	4
7	RTS	7	Gray	7	7	7	7	Gray	8
8	CTR	8	White/ Green	8	8	8	8	White/ Green	7
9	Do not assign	9	Green	9	9	9	9	Green	9
Hsg.	External shield	Hsg.	External shield	Hsg.	Hsg.	Hsg.	Hsg.	External shield	Hsg.

Tables and overviews

19.2 Connector pin layout and connection cables for data interfaces

Non-HEIDENHAIN devices

The connector layout of a non-HEIDENHAIN device may substantially differ from that of a HEIDENHAIN device.

It depends on the unit and the type of data transfer. The table below shows the connector pin layout on the adapter block.

Adapter block 363987-02		Conn. cable 366964-xx		
Female	Male	Female	Color	Female
1	1	1	Red	1
2	2	2	Yellow	3
3	3	3	White	2
4	4	4	Brown	6
5	5	5	Black	5
6	6	6	Violet	4
7	7	7	Gray	8
8	8	8	White/ Green	7
9	9	9	Green	9
Hsg.	Hsg.	Hsg.	Ext. shield	Hsg.

Ethernet interface RJ45 socket

Maximum cable length:

- Unshielded: 100 m
- Shielded: 400 m

Pin	Signal	Description
1	TX+	Transmit Data
2	TX-	Transmit Data
3	REC+	Receive Data
4	Vacant	
5	Vacant	
6	REC-	Receive Data
7	Vacant	
8	Vacant	

19.3 Technical Information

Explanation of symbols

- Default
- Axis option
- 1 Advanced Function Set 1
- 2 Advanced Function Set 2

User functions

Short description	<ul style="list-style-type: none"> ■ 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 shaft speed control
Short description	<ul style="list-style-type: none"> ■ Basic version: 3 axes plus closed-loop spindle □ 1. Additional axis for 4 axes plus closed-loop spindle □ 2. Additional axis for 5 axes plus closed-loop spindle
Program entry	In HEIDENHAIN conversational and DIN/ISO
Position entry	<ul style="list-style-type: none"> ■ 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	<ul style="list-style-type: none"> ■ Tool radius in the working plane and tool length ■ Radius compensated contour look ahead for up to 99 blocks (M120) 2 Three-dimensional tool-radius compensation for changing tool data without having to recalculate an existing program
Tool tables	Multiple tool tables with any number of tools
Constant contour speed	<ul style="list-style-type: none"> ■ With respect to the path of the tool center ■ With respect to the cutting edge
Parallel operation	Creating a program with graphical support while another program is being run
3-D machining (Advanced Function Set 2)	<ul style="list-style-type: none"> 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 point. (TCPM = Tool Center Point Management) 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)	<ul style="list-style-type: none"> 1 Programming of cylindrical contours as if in two axes 1 Feed rate in distance per minute
Contour elements	<ul style="list-style-type: none"> ■ Straight line ■ Chamfer ■ Circular path

User functions

	<ul style="list-style-type: none"> ■ Circle center ■ Circle radius ■ Tangentially connected arc ■ Corner rounding
Approaching and departing the contour	<ul style="list-style-type: none"> ■ Via straight line: tangential or perpendicular ■ Via circular arc
FK free contour programming	<ul style="list-style-type: none"> ■ FK free contour programming in HEIDENHAIN conversational format with graphic support for workpiece drawings not dimensioned for NC
Program jumps	<ul style="list-style-type: none"> ■ Subprograms ■ Program section repeats ■ Any desired program as subprogram
Fixed cycles	<ul style="list-style-type: none"> ■ 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 tool builder) can also be integrated
Coordinate transformation	<ul style="list-style-type: none"> ■ Datum shift, rotation, mirroring ■ Scaling factor (axis-specific) 1 Tilting the working plane (Advanced Function Set 1)
Q parameters Programming with variables	<ul style="list-style-type: none"> ■ Mathematical functions: =, +, -, *, sin α, cos α, root ■ Logical operations (=, \neq, <, >) ■ 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 param.
Programming aids	<ul style="list-style-type: none"> ■ Calculator ■ Complete list of all current error messages ■ Context-sensitive help function for error messages ■ Graphic support for the programming of cycles ■ Comment blocks in the NC program

User functions

Teach-In	■	Actual positions can be transferred directly into the NC program
Program verification graphics Display modes	■	Graphic simulation before program run, even while another program is being run
	■	Plan view / projection in 3 planes / 3-D view / 3-D line graphic
	■	Magnification of details
Programming graphics	■	In the Programming mode, the contour of the NC blocks is drawn on screen while they are being entered (2-D pencil-trace graphics), even while another program is running
Program-run graphics Display modes	■	Graphic simulation of real-time machining in plan view / projection in 3 planes / 3-D view
Machining time	■	Calculating the machining time in the Test Run mode of operation
	■	Display of the current machining time in the Program Run operating modes
Contour, returning to	■	Mid-program startup in any block in the program, returning the tool to the calculated nominal position to continue machining
	■	Program interruption, contour departure and return
Datum tables	■	Multiple datum tables, for storing workpiece-related datums
Touch probe cycles	■	Calibrating the touch probe
	■	Compensation of workpiece misalignment, manual or automatic
	■	Datum setting, manual or automatic
	■	Automatically measuring workpieces
	■	Cycles for automatic tool measurement
	■	Cycles for automatic kinematics measurement

Specifications

Components	<ul style="list-style-type: none"> ■ Operating panel ■ TFT color flat-panel display with soft keys
Program memory	<ul style="list-style-type: none"> ■ Minimum 21 GB
Input resolution and display step	<ul style="list-style-type: none"> ■ As fine as 0.1 µm for linear axes ■ As fine as 0.01 µm for linear axes (with option 23) ■ Up to 0.0001° for rotary axes ■ Up to 0.000 01° for rotary axes (with option 23)
Input range	<ul style="list-style-type: none"> ■ Maximum 999 999 999 mm or 999 999 999°
Interpolation	<ul style="list-style-type: none"> ■ Linear in 4 axes ■ Circular in 2 axes ■ Helical: superimposition of circular and straight paths
Block processing time 3-D straight line without radius compensation	<ul style="list-style-type: none"> ■ 0.5 ms
Axis feedback control	<ul style="list-style-type: none"> ■ Position loop resolution: Signal period of the position encoder/1024 ■ Cycle time of position controller: 3 ms ■ Cycle time of speed controller: 200 µs
Range of traverse	<ul style="list-style-type: none"> ■ Maximum 100 m (3937 inches)
Spindle speed	<ul style="list-style-type: none"> ■ Maximum 100 000 rpm (analog speed command signal)
Error compensation	<ul style="list-style-type: none"> ■ Linear and nonlinear axis error, backlash, reversal peaks during circular movements, thermal expansion ■ Static friction
Data interfaces	<ul style="list-style-type: none"> ■ One each RS-232-C /V.24 max. 115 kilobaud ■ Expanded interface with LSV-2 protocol for external operation of the TNC over the interface with HEIDENHAIN software TNCremo ■ Ethernet interface 1000 Base T ■ 3 x USB 2.0
Ambient temperature	<ul style="list-style-type: none"> ■ Operation: 0 °C to +45°C ■ Storage: -30°C to +70°C

Accessories**Electronic Handwheels**

- One HR 410 portable handwheel, or
- One HR 550 FS portable wireless handwheel with display or
- One HR 520 portable handwheel with display, or
- One HR 420 portable handwheel with display or
- One HR 130 panel-mounted handwheel, or
- Up to three HR 150 panel-mounted handwheels via HRA 110 handwheel adapter

Touch probes

- TS 220: Triggering 3-D touch probe with cable connection
- TS 440: 3-D touch trigger probe with infrared transmission
- TS 444: Battery-free 3-D touch trigger probe with infrared transmission
- TS 640: 3-D touch trigger probe with infrared transmission
- TS 740: High-precision 3-D touch trigger probe with infrared transmission
- TT 140: 3-D touch trigger probe for tool measurement
- TT 449: 3-D touch trigger probe for tool measurement with infrared transmission

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 transformations:

Tilting the working plane

Interpolation:

Circle in 3 axes with tilted working plane (spatial arc)

Advanced Function Set 2 (option 9)**Expanded functions Group 2****3-D machining:**

- Motion control with minimum jerk
- 3-D tool compensation through surface normal vectors
- Using the electronic handwheel to change the angle of the swivel head during program run without affecting the position of the tool point. (TCPM = Tool Center Point Management)
- Keeping the tool normal to the contour
- Tool radius compensation perpendicular to traversing direction and tool direction

Interpolation:

Linear in 5 axes (subject to export permit)

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	<ul style="list-style-type: none"> ■ The machine manufacturer defines objects to be monitored ■ Warning in Manual operation ■ Program interrupt in Automatic operation ■ Includes monitoring of 5-axis movements
-------------------------------------	--

DXF Converter (Option #42)

DXF converter	<ul style="list-style-type: none"> ■ Supported DXF format: AC1009 (AutoCAD R12) ■ Adoption of contours and point patterns ■ Simple and convenient specification of reference points ■ Select graphical features of contour sections from conversational programs
----------------------	--

Adaptive Feed Control – AFC (Option #45)

Adaptive Feed Control	<ul style="list-style-type: none"> ■ 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
------------------------------	--

KinematicsOpt (Option #48)

Optimizing the machine kinematics	<ul style="list-style-type: none"> ■ Backup/restore active kinematics ■ Test active kinematics ■ Optimize active kinematics
--	--

Mill-Turning (Option #50)

Milling and turning modes	Functions: <ul style="list-style-type: none"> ■ Switching between Milling/Turning mode of operation ■ Constant surface speed ■ Tool-tip radius compensation ■ Turning cycles
----------------------------------	---

Extended Tool Management (Option #93)

Extended tool management	Python-based
---------------------------------	--------------

Spindle Synchronism (Option #131)

Spindle synchronization	Synchronization of milling spindle and turning spindle
--------------------------------	--

Remote Desktop Manager (Option #133)

Remote operation of external computer units	<ul style="list-style-type: none"> ■ Windows on a separate computer unit ■ Incorporated in the TNC interface
--	--

Synchronizing Functions (Option #135)

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

Cross Talk Compensation – CTC (Option #141)

Compensation of axis couplings	<ul style="list-style-type: none"> ■ Determination of dynamically caused position deviation through axis acceleration ■ Compensation of TCP (Tool Center Point)
---------------------------------------	--

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

Input format and unit of TNC functions

Positions, coordinates, circle radii, chamfer lengths	-99 999.9999 to +99 999.9999 (5, 4: places before the decimal point, places after the decimal point) [mm]
Tool numbers	0 to 32 767.9 (5, 1)
Tool names	32 characters, enclosed by quotation marks with TOOL CALL . Permitted special characters: #, \$, %, &, -
Delta values for tool compensation	-99.9999 to +99.9999 (2, 4) [mm]
Spindle speeds	0 to 99 999.999 (5, 3) [rpm]
Feed rates	0 to 99 999.999 (5.3) [mm/min] or [mm/tooth] or [mm/rev]
Dwell time in Cycle 9	0 to 3600.000 (4, 3) [s]
Thread pitch in various cycles	-99.9999 to +99.9999 (2, 4) [mm]
Angle of spindle orientation	0 to 360.0000 (3, 4) [°]
Angle for polar coordinates, rotation, tilting the working plane	-360.0000 to +360.0000 (3, 4) [°]
Polar coordinate angle for helical interpolation (CP)	-5 400.0000 to 5 400.0000 (4, 4) [°]
Datum numbers in Cycle 7	0 to 2999 (4, 0)
Scaling factor in Cycles 11 and 26	0.000001 to 99.999999 (2, 6)
Miscellaneous functions M	0 to 999 (4, 0)
Q parameter numbers	0 to 1999 (4, 0)
Q parameter values	-99 999.9999 to +99 999.9999 (9, 6)
Surface-normal vectors N and T with 3-D compensation	-9.99999999 to +9.99999999 (1, 8)
Labels (LBL) for program jumps	0 to 999 (5, 0)
Labels (LBL) for program jumps	Any text string in quotes (" ")
Number of program section repeats REP	1 to 65 534 (5, 0)
Error number with Q parameter function FN14	0 to 1199 (4, 0)

Tables and overviews

19.4 Overview tables

19.4 Overview tables

Fixed cycles

Cycle number	Cycle designation	DEF active	CALL active
7	Datum shift	■	
8	Mirror image	■	
9	Dwell time	■	
10	Rotation	■	
11	Scaling factor	■	
12	Program call	■	
13	Spindle orientation	■	
14	Contour definition	■	
19	Tilting the working plane	■	
20	Contour data SL II	■	
21	Pilot drilling SL II		■
22	Rough out SL II		■
23	Floor finishing SL II		■
24	Side finishing SL II		■
25	Contour train		■
26	Axis-specific scaling	■	
27	Cylinder surface		■
28	Cylindrical surface slot		■
29	Cylinder surface ridge		■
32	Tolerance	■	
200	Drilling		■
201	Reaming		■
202	Boring		■
203	Universal drilling		■
204	Back boring		■
205	Universal pecking		■
206	Tapping with a floating tap holder, new		■
207	Rigid tapping, new		■
208	Bore milling		■
209	Tapping with chip breaking		■
220	Polar pattern	■	
221	Cartesian pattern	■	
230	Multipass milling		■
231	Ruled surface		■
232	Face milling		■

Cycle number	Cycle designation	DEF active	CALL active
233	Face milling (selectable machining direction, consider the sides)		■
240	Centering		■
241	Single-lip deep-hole drilling		■
247	Datum setting	■	
251	Rectangular pocket (complete machining)		■
252	Circular pocket (complete machining)		■
253	Slot milling		■
254	Circular slot		■
256	Rectangular stud (complete machining)		■
257	Circular stud (complete machining)		■
262	Thread milling		■
263	Thread milling/countersinking		■
264	Thread drilling/milling		■
265	Helical thread drilling/milling		■
267	Outside thread milling		■
275	Trochoidal slot		■

Miscellaneous functions

M	Effect	Effective at block...	Start	End	Page
M0	Program STOP/Spindle STOP/Coolant OFF			■	369
M1	Optional program run STOP/Spindle STOP/Coolant OFF			■	610
M2	Stop program/Spindle STOP/Coolant OFF/ CLEAR status display (depending on machine parameter)/Return jump to block 1			■	369
M3	Spindle ON clockwise		■		369
M4	Spindle ON counterclockwise		■		
M5	Spindle STOP			■	
M6	Tool change/STOP program run (depending on machine parameter)/Spindle STOP			■	369
M8	Coolant on		■		369
M9	Coolant off			■	
M13	Spindle ON clockwise /coolant ON		■		369
M14	Spindle ON counterclockwise/coolant on		■		
M30	Same function as M2			■	369
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		■		370
M92	Within the positioning block: Coordinates are referenced to position defined by machine tool builder, such as tool change position		■		370
M94	Reduce the rotary axis display to a value below 360°		■		463

Tables and overviews

19.4 Overview tables

M	Effect	Effective at block...	Start	End	Page
M97	Machine small contour steps			■	373
M98	Machine open contours completely			■	374
M99	Blockwise cycle call			■	Cycles Manual
M101	Automatic tool change with replacement tool if maximum tool life has expired			■	188
M102	Reset M101			■	
M107	Suppress error message for replacement tools with oversize			■	188
M108	Reset M107			■	
M109	Constant contouring speed at cutting edge (feed rate increase and reduction)	■			377
M110	Constant contouring speed at cutting edge (only feed rate reduction)	■			
M111	Reset M109/M110			■	
M116	Feed rate in mm/min on rotary axes	■			461
M117	Reset M116			■	
M118	Superimpose handwheel positioning during program run	■			380
M120	Pre-calculate the radius-compensated contour (LOOK AHEAD)	■			378
M126	Shorter-path traverse of rotary axes:	■			462
M127	Reset M126			■	
M128	Maintaining the position of the tool tip when positioning with tilted axes (TCPM)	■			464
M129	Reset M128			■	
M130	Within the positioning block: Points are referenced to the untilted coordinate system	■			372
M138	Selection of tilted axes	■			467
M140	Retraction from the contour in the tool-axis direction	■			382
M143	Delete basic rotation	■			385
M144	Compensating the machine's kinematic configuration for ACTUAL/ NOMINAL positions at end of block	■			468
M145	Reset M144			■	
M141	Suppress touch probe monitoring	■			384
M148	Automatically retract tool from the contour at an NC stop	■			386
M149	Reset M148			■	

19.5 Functions of the TNC 640 and the iTNC 530 compared

Comparison: Specifications

Function	TNC 640	iTNC 530
Axes	18 maximum	18 maximum
Input resolution and display step:		
■ Linear axes	■ 0.1 μm, 0.01 μm with option 23	■ 0.1 μm
■ Rotary axes	■ 0.001°, 0.00001° with option 23	■ 0.0001°
Control loops for high-frequency spindles and torque/linear motors	With option 49	With option 49
Display	19-inch TFT color flat-panel display or	19-inch TFT color flat-panel display or 15.1-inch TFT color flat-panel display
Memory media for NC, PLC programs and system files	Hard disk or SSDR solid state disk	Hard disk or SSDR solid state disk
Program memory for NC programs	> 21 GB	> 21 GB
Block processing time	0.5 ms	0.5 ms
HeROS operating system	yes	yes
Interpolation:		
■ Straight line	■ 5 axes	■ 5 axes
■ Circle	■ 3 axes	■ 3 axes
■ Helix	■ yes	■ yes
■ Spline	■ no	■ Yes with option 9
Hardware	modular in electrical cabinet	Modular in electrical cabinet

Comparison: Data interfaces

Function	TNC 640	iTNC 530
Gigabit Ethernet 1000BaseT	X	X
RS-232-C/V.24 serial interface	X	X
RS-422/V.11 serial interface	-	X
USB interface	X	X

Tables and overviews

19.5 Functions of the TNC 640 and the iTNC 530 compared

Comparison: Accessories

Function	TNC 640	iTNC 530
Electronic handwheels		
■ HR 410	■ X	■ X
■ HR 420	■ X	■ X
■ HR 520/530/550	■ X	■ X
■ HR 130	■ X	■ X
■ HR 150 via HRA 110	■ X	■ X
Touch probes		
■ TS 220	■ X	■ X
■ TS 440	■ X	■ X
■ TS 444	■ X	■ X
■ TS 449 / TT 449	■ X	■ X
■ TS 640	■ X	■ X
■ TS 740	■ X	■ X
■ TT 130 / TT 140	■ X	■ X
Industrial PC IPC 61xx	–	X

Comparison: PC software

Function	TNC 640	iTNC 530
Programming station software	Available	Available
TNCremoNT for data transfer with TNCbackup for data backup	Available	Available
TNCremoPlus data transfer software with "live" screen	Available	Available
RemoTools SDK 1.2: Function library for developing your own applications for communicating with HEIDENHAIN controls	Limited functionality available	Available
virtualTNC: Control component for virtual machines	Not available	Available
ConfigDesign: Software for configuring the control	Available	Not available
TeleService: Software for remote diagnostics and maintenance	Available	Available

Functions of the TNC 640 and the iTNC 530 compared 19.5

Comparison: Machine-specific functions

Function	TNC 640	iTNC 530
Switching the traverse range	Function available	Function available
Central drive (1 motor for multiple machine axes)	Function available	Function available
C-axis operation (spindle motor drives rotary axis)	Function available	Function available
Automatic exchange of milling head	Function available	Function available
Support of angle heads	Function not available	Function available
Balluf tool identification	Function available (with Python)	Function available
Management of multiple tool magazines	Function available	Function available
Expanded tool management via Python	Function available	Function available

Comparison: User Functions

Function	TNC 640	iTNC 530
Program entry		
■ HEIDENHAIN conversational	■ X	■ X
■ DIN/ISO	■ X	■ X
■ With smarT.NC	■ –	■ X
■ With ASCII editor	■ X, directly editable	■ X, editable after conversion
Position entry		
■ Nominal positions for lines and arcs in Cartesian coordinates	■ X	■ X
■ Nominal positions for lines and arcs in polar coordinates	■ X	■ X
■ Incremental or absolute dimensions	■ X	■ X
■ Display and entry in mm or inches	■ X	■ X
■ Set the last tool position as pole (empty CC block)	■ X (error message if pole transfer is ambiguous)	■ X
■ Surface normal vectors (LN)	■ X	■ X
■ Spline blocks (SPL)	■ –	■ X, with option 9

Tables and overviews

19.5 Functions of the TNC 640 and the iTNC 530 compared

Function	TNC 640	iTNC 530
Tool compensation		
■ In the working plane and tool length	■ X	■ X
■ Radius compensated contour look ahead for up to 99 blocks	■ X	■ X
■ Three-Dimensional Tool Radius Compensation	■ X, with option 9	■ X, with option 9
Tool table		
■ Central storage of tool data	■ X	■ X
■ Multiple tool tables with any number of tools	■ X	■ X
■ Flexible management of tool types	■ X	■ –
■ Filtered display of selectable tools	■ X	■ –
■ Sorting function	■ X	■ –
■ Column names	■ Sometimes with _	■ Sometimes with -
■ Copy function: Overwriting relevant tool data	■ X	■ X
■ Form view	■ Switchover with split-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	–
Creating tool-usage file, checking the availability	X	X
Cutting data calculation Automatic calculation of spindle speed and feed rate	Simple cutting data calculator	Using technology tables
Define any tables	<ul style="list-style-type: none"> ■ Freely definable tables (.TAB files) ■ Reading and writing with FN functions ■ Definable via config. data ■ Table names must start with a letter ■ Reading and writing with SQL functions 	<ul style="list-style-type: none"> ■ Freely definable tables (.TAB files) ■ Reading and writing with FN functions

Functions of the TNC 640 and the iTNC 530 compared 19.5

Function	TNC 640	iTNC 530
Constant contouring speed relative to the path of the tool center or relative to the tool's cutting edge	X	X
Parallel operation: Creating programs while another program is being run	X	X
Programming of counter axes	X	X
Tilting the working plane (Cycle 19, PLANE function)	X, option 8	X, option 8
Machining with rotary tables		
■ Programming of cylindrical contours as if in two axes		
■ Cylinder Surface (Cycle 27)	■ X, option 8	■ X, option 8
■ Cylinder Surface Slot (Cycle 28)	■ X, option 8	■ X, option 8
■ Cylinder Surface Ridge (Cycle 29)	■ X, option 8	■ X, option 8
■ Cylinder Surface External Contour (Cycle 39)	■ –	■ X, option 8
■ Feed rate in mm/min or rev/min	■ X, option 8	■ X, option 8
Traverse in tool-axis direction		
■ Manual operation (3-D ROT menu)	■ X	■ X, FCL2 function
■ During program interruption	■ X	■ X
■ With handwheel superimpositioning	■ X	■ X, option #44
Approaching and departing the contour: Via a straight line or arc	X	X
Entry of feed rates:		
■ F (mm/min), rapid traverse FMAX	■ X	■ X
■ FU (feed per revolution mm/rev)	■ X	■ X
■ FZ (tooth feed rate)	■ X	■ X
■ FT (time in seconds for path)	■ –	■ X
■ FMAXT (only for active rapid traverse pot: time in seconds for path)	■ –	■ X
FK free contour programming		
■ Programming for workpiece drawings not dimensioned for NC programming	■ X	■ X
■ Conversion of FK program to conversational dialog	■ –	■ X
Program jumps:		
■ Maximum number of label numbers	■ 9999	■ 1000
■ Subprograms	■ X	■ X
■ Nesting depth for subprograms	■ 20	■ 6
■ Program section repeats	■ X	■ X
■ Any desired program as subroutine	■ X	■ X

19.5 Functions of the TNC 640 and the iTNC 530 compared

Function	TNC 640	iTNC 530
Q parameter programming:		
■ Standard mathematical functions	■ X	■ X
■ Formula entry	■ X	■ X
■ String processing	■ X	■ X
■ Local Q parameters QL	■ X	■ X
■ Nonvolatile Q parameters QR	■ X	■ X
■ Changing parameters during program interruption	■ X	■ X
■ FN15:PRINT	■ –	■ X
■ FN25:PRESET	■ –	■ X
■ FN26:TABOPEN	■ X	■ X
■ FN27:TABWRITE	■ X	■ X
■ FN28:TABREAD	■ X	■ X
■ FN29: PLC LIST	■ X	■ –
■ FN31: RANGE SELECT	■ –	■ X
■ FN32: PLC PRESET	■ –	■ X
■ FN37:EXPORT	■ X	■ –
■ FN38: SEND	■ X	■ X
■ Saving file externally with FN16	■ X	■ X
■ FN16 formatting: Left-aligned, right-aligned, string lengths	■ X	■ X
■ Writing to LOG file with FN16	■ X	■ –
■ Displaying parameter contents in the additional status display	■ X	■ –
■ Displaying parameter contents during programming (Q-INFO)	■ X	■ X
■ SQL functions for writing and reading tables	■ X	■ –

Functions of the TNC 640 and the iTNC 530 compared 19.5

Function	TNC 640	iTNC 530
Graphic support		
■ 2-D programming graphics	■ X	■ X
■ REDRAW function	■ –	■ X
■ Show grid lines as the background	■ X	■ –
■ 3-D line graphics	■ X	■ X
■ Test graphics (plan view, projection in 3 planes, 3-D view)	■ X	■ X
■ High-resolution view	■ X	■ X
■ Tool display	■ X	■ X
■ Adjusting the simulation speed	■ X	■ X
■ Coordinates of line intersection for projection in 3 planes	■ –	■ X
■ Expanded zoom functions (mouse operation)	■ X	■ X
■ Displaying frame for workpiece blank	■ X	■ X
■ Displaying the depth value in plan view during mouse-over	■ –	■ X
■ Targeted stop of test run (STOP AT N)	■ –	■ X
■ Consider tool change macro	■ –	■ X
■ Program run graphics (plan view, projection in 3 planes, 3-D view)	■ X	■ X
■ High-resolution view	■ X	■ X

Tables and overviews

19.5 Functions of the TNC 640 and the iTNC 530 compared

Function	TNC 640	iTNC 530
Datum tables: for storing workpiece-related datums	X	X
Preset table: for saving reference points (presets)	X	X
Pallet management		
■ Support of pallet files	■ X	■ X
■ Tool-oriented machining	■ –	■ X
■ Pallet preset table: for managing pallet datums	■ –	■ X
Returning to the contour		
■ With mid-program startup	■ X	■ X
■ After program interruption	■ X	■ X
Autostart function	X	X
Actual position capture: Actual positions can be transferred to the NC program	X	X
Enhanced file management		
■ Creating multiple directories and subdirectories	■ X	■ X
■ Sorting function	■ X	■ X
■ Mouse operation	■ X	■ X
■ Selection of target directory by soft key	■ X	■ X
Programming aids:		
■ Help graphics for cycle programming	■ X	■ X
■ Animated help graphics when PLANE/PATTERN DEF function is selected	■ –	■ X
■ Help graphics for PLANE/PATTERN DEF	■ X	■ X
■ Context-sensitive help function for error messages	■ X	■ X
■ TNCguide: Browser-based help system	■ X	■ X
■ Context-sensitive call of help system	■ X	■ X
■ Calculator	■ X (scientific)	■ X (standard)
■ Comment blocks in NC program	■ X	■ X
■ Structure blocks in NC program	■ X	■ X
■ Structure view in test run	■ –	■ X
Dynamic Collision Monitoring (DCM):		
■ Collision monitoring in Automatic operation	■ X, option #40	■ X, option #40
■ Collision monitoring in Manual operation	■ X, option #40	■ X, option #40
■ Graphic depiction of the defined collision objects	■ X, option #40	■ X, option #40
■ Collision checking in the Test Run mode	■ –	■ X, option #40
■ Fixture monitoring	■ –	■ X, option #40
■ Tool carrier management	■ –	■ X, option #40

Functions of the TNC 640 and the iTNC 530 compared 19.5

Function	TNC 640	iTNC 530
CAM support:		
■ Loading of contours from DXF data	■ X, option #42	■ X, option #42
■ Loading of machining positions from DXF data	■ X, option 42	■ 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
■ Setting the system time	■ X	■ X
■ Select the axes for actual position capture	■ –	■ X
■ Definition of traverse range limits	■ X	■ X
■ Restricting external access	■ X	■ X
■ Switching the kinematics	■ X	■ X
Calling fixed cycles:		
■ With M99 or M89	■ X	■ X
■ With CYCL CALL	■ X	■ X
■ With CYCL CALL PAT	■ X	■ X
■ With CYC CALL POS	■ X	■ X
Special functions:		
■ Creating backward programs	■ –	■ X
■ Datum shift with TRANS DATUM	■ X	■ X
■ Adaptive Feed Control AFC	■ X, option #45	■ X, option #45
■ Global definition of cycle parameters: GLOBAL DEF	■ X	■ X
■ Pattern definition with PATTERN DEF	■ X	■ X
■ Definition and execution of point tables	■ X	■ X
■ Simple contour formula CONTOUR DEF	■ X	■ X
Functions for large molds and dies:		
■ Global program settings (GS)	■ –	■ X, option #44
■ Expanded M128: FUNCTION TCPM	■ X	■ X
Status displays:		
■ Positions, spindle speed, feed rate	■ X	■ X
■ Larger depiction of position display, Manual operation	■ X	■ X
■ Additional status display, form view	■ X	■ X
■ Display of handwheel traverse when machining with handwheel superimposition	■ X	■ X
■ Display of distance-to-go in a tilted system	■ –	■ X

Tables and overviews

19.5 Functions of the TNC 640 and the iTNC 530 compared

Function	TNC 640	iTNC 530
■ Dynamic display of Q-parameter contents, definable number ranges	■ X	■ –
■ OEM-specific additional status display via Python	■ X	■ X
■ Graphic display of residual run time	■ –	■ X
Individual color settings of user interface	–	X

Comparator: Cycles

Cycle	TNC 640	iTNC 530
1, Pecking	X	X
2, Tapping	X	X
3, Slot milling	X	X
4, Pocket milling	X	X
5, Circular pocket	X	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, Program call	X	X
13, Spindle orientation	X	X
14, Contour definition	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, tapping (controlled spindle)	X	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, Cylinder surface ridge	X, option 8	X, option 8
30, run 3-D data	–	X
32, tolerance with HSC mode and TA	X	X

Functions of the TNC 640 and the iTNC 530 compared 19.5

Cycle	TNC 640	iTNC 530
39, Cylinder surface external 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
205, Universal pecking	X	X
206, Tapping with floating tap holder	X	X
207, Rigid tapping	X	X
208, Bore milling	X	X
209, Tapping with chip breaking	X	X
210, Slot with reciprocating plunge	X	X
211, Circular slot	X	X
212, Rectangular pocket finishing	X	X
213, Rectangular stud finishing	X	X
214, Circular pocket finishing	X	X
215, Circular stud finishing	X	X
220, Polar pattern	X	X
221, Cartesian pattern	X	X
225, Engraving	X	X
230, Multipass milling	X	X
231, Ruled surface	X	X
232, Face milling	X	X
233, Face milling, new	X	–
239, Ascertain the load	X, option 143	–
240, Centering	X	X
241, single-lip deep-hole drilling	X	X
247, Datum setting	X	X
251, Rectangular pocket (complete)	X	X
252, Circular pocket (complete)	X	X
253, Slot milling (complete)	X	X
254, Circular slot (complete)	X	X
256, Rectangular stud (complete)	X	X
257, Circular stud (complete)	X	X
262, Thread milling	X	X
263, Thread milling/counter sinking	X	X
264, Thread drilling/milling	X	X
265, Helical thread drilling/milling	X	X
267, outside thread milling	X	X

19.5 Functions of the TNC 640 and the iTNC 530 compared

Cycle	TNC 640	iTNC 530
270, contour train data for defining the behavior of Cycle 25	X	X
275, trochoidal milling	X	X
276, 3-D contour train	–	X
290, Interpolation turning	–	X, option 96
291, Interpolation turning, coupling	X, option 96	–
292, Interpolation turning, contour	X, option 96	–
800, Adapt rotary coordinate system	X	–
801, Reset rotary coordinate system	X	–
810, Turn contour, longitudinal	X	–
811, Turn shoulder, longitudinal	X	–
812, Turn shoulder, longitudinal extended	X	–
813, Turn, longitudinal plunge	X	–
814, Turn, longitudinal plunge extended	X	–
815, Turn contour-parallel	X	–
820, Turn contour, transverse	X	–
821, Turn shoulder face	X	–
822, Turn shoulder face extended	X	–
823, Turn, transverse plunge	X	–
824, Turn, transverse plunge extended	X	–
830, Thread, contour-parallel	X	–
831, Thread, longitudinal	X	–
832, Thread, extended	X	–
840, Recessing contour, radial	X	–
841, Recessing simple, radial	X	–
842, Recessing, radial extended	X	–
850, Recessing contour, axial	X	–
851, Recessing simple axial	X	–
852, Recessing axial extended	X	–
860, Recessing contour, radial	X	–
861, Recessing, radial	X	–
862, Recessing, radial extended	X	–
870, Recessing contour, axial	X	–
871, Recessing, axial	X	–
872, Recessing, axial extended	X	–
880, Gear wheel hobbing	X, option 131	–
892, Check imbalance	X	–

Functions of the TNC 640 and the iTNC 530 compared 19.5

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	Spindle ON clockwise	X	X
M04	Spindle ON counterclockwise		
M05	Spindle STOP		
M06	Tool change/Stop program run (machine-dependent function)/ Spindle STOP	X	X
M08	Coolant on	X	X
M09	Coolant off		
M13	Spindle ON clockwise /coolant ON	X	X
M14	Spindle ON counterclockwise/coolant on		
M30	Same function as M02	X	X
M89	Vacant miscellaneous function or cycle call, modally effective (machine-dependent 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 position defined by machine tool builder, such as 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	Automatic tool change with replacement tool if maximum tool life has expired	X	X
M102	Reset M101		
M103	Reduce feed rate during plunging to factor F (percentage)	X	X
M104	Reactivate most recently set datum	– (recommended: Cycle 247)	X
M105	Machining with second k_v factor	–	X
M106	Machining with first k_v factor		
M107	Suppress error message for replacement tools with oversize	X	X
M108	Reset M107		
M109	Constant contouring speed at cutting edge (feed rate increase and reduction)	X	X
M110	Constant contouring speed at cutting edge (only feed rate reduction)		
M111	Reset M109/M110		

Tables and overviews

19.5 Functions of the TNC 640 and the iTNC 530 compared

M	Effect	TNC 640	iTNC 530
M112 M113	Enter contour transition between two contour elements Reset M112	– (recommended: Cycle 32)	X
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 with 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	Exact stop at nontangential 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

Comparison: Touch probe cycles in the Manual Operation and El. Handwheel modes

Cycle	TNC 640	iTNC 530
Touch-probe table for managing 3-D touch probes	X	–
Calibrating the effective length	X	X
Calibrating the effective radius	X	X
Measuring a basic rotation using a line	X	X
Set the datum in any axis	X	X
Setting a corner as datum	X	X
Setting a circle center as datum	X	X
Setting a center line as datum	X	X
Measuring a basic rotation using two holes/cylindrical studs	X	X
Setting the datum using four holes/cylindrical studs	X	X
Setting the circle center using three holes/cylindrical studs	X	X
Support of mechanical touch probes by manually capturing the current position	By soft key	By hard key
Writing measured values in preset table	X	X
Writing measured values in datum tables	X	X

19.5 Functions of the TNC 640 and the iTNC 530 compared

Comparison: Touch probe cycles for automatic workpiece inspection

Cycle	TNC 640	iTNC 530
0, reference plane	X	X
1, polar datum	X	X
2, calibrating TS	–	X
3, measuring	X	X
4, measuring in 3-D	X	X
9, calibrating TS length	–	X
30, calibrating TT	X	X
31, measuring tool length	X	X
32, measuring tool radius	X	X
33, measuring tool length and radius	X	X
400, basic rotation	X	X
401, basic rotation from two holes	X	X
402, basic rotation from two studs	X	X
403, compensating a basic rotation via a rotary axis	X	X
404, setting a basic rotation	X	X
405, compensating workpiece misalignment by rotating the C axis	X	X
408, slot center datum	X	X
409, ridge center datum	X	X
410, datum from inside of rectangle	X	X
411, datum from outside of rectangle	X	X
412, datum from inside of circle	X	X
413, datum from outside of circle	X	X
414, datum at outside corner	X	X
415, datum at inside corner	X	X
416, datum at circle center	X	X
417, datum in touch probe axis	X	X
418, datum at center of 4 holes	X	X
419, datum in one axis	X	X
420, measuring an angle	X	X
421, measuring a hole	X	X
422, measuring a circle from outside	X	X
423, measuring a rectangle from inside	X	X
424, measuring a rectangle from outside	X	X
425, measuring inside width	X	X
426, measuring a ridge from outside	X	X
427, boring	X	X
430, measuring a bolt hole circle	X	X

Functions of the TNC 640 and the iTNC 530 compared 19.5

Cycle	TNC 640	iTNC 530
431, measuring a plane	X	X
440, measuring an axis shift	–	X
441, Rapid probing (on TNC 640 partly possible with touch probe table)	–	X
450, saving the kinematics	X, option 48	X, option 48
451, measuring the kinematics	X, option 48	X, option 48
452, preset compensation	X, option 48	X, option 48
460, calibrating a TS on a sphere	X	X
461, calibrate TS length	X	X
462, calibration in a ring	X	X
463, calibration on stud	X	X
480, calibrating a TT	X	X
481, measuring/inspecting the tool length	X	X
482, measuring/inspecting the tool radius	X	X
483, measuring/inspecting the tool length and radius	X	X
484, calibrating the infrared TT	X	X

Comparison: Differences in programming

Function	TNC 640	iTNC 530
Switching the operating mode while a block is being edited	Permitted	Permitted
File handling:		
■ Save file function	■ Available	■ Available
■ Save file as function	■ Available	■ Available
■ Discard changes	■ Available	■ Available
File management:		
■ Mouse operation	■ Available	■ Available
■ Sorting function	■ Available	■ Available
■ Entry of name	■ Opens the Select file pop-up window	■ Synchronizes the cursor
■ Support of shortcuts	■ Not available	■ Available
■ Favorites management	■ Not available	■ Available
■ Configuration of column structure	■ Not available	■ Available
■ Soft-key arrangement	■ Slightly different	■ Slightly different
Skip block function	Available	Available
Selecting a tool from the table	Selection via split-screen menu	Selection in a pop-up window

19.5 Functions of the TNC 640 and the iTNC 530 compared

Function	TNC 640	iTNC 530
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 TNC 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 TNC 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 TNC 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 TNC shows the last active soft-key row
Pressing the END hard key while the CYCLE DEF and TOUCH PROBE menus are active	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	Error message Key non-functional
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

Functions of the TNC 640 and the iTNC 530 compared 19.5

Function	TNC 640	iTNC 530
Datum table:		
■ Sorting function by values within an axis	■ Available	■ Not available
■ Resetting the table	■ Available	■ Not available
■ Hiding axes that are not present	■ Available	■ Available
■ Switching the list/form view	■ Switchover via split-screen 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 end of table. Line with value 0 in all columns is inserted
■ Transfer of actual position values in individual axis to the datum table per keystroke	■ Not available	■ Available
■ Transfer of actual position values in all active axes to the datum table per keystroke	■ Not available	■ Available
■ Using a key to capture the last positions measured by TS	■ 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

19.5 Functions of the TNC 640 and the iTNC 530 compared

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

Functions of the TNC 640 and the iTNC 530 compared 19.5

Function	TNC 640	iTNC 530
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 find function with the up/down arrow keys when highlight is on a block	Works with max. 9999 blocks, can be set via config 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	■ If error messages occur, the cursor is on the CYCL CALL block in the main program	■ If error messages occur, the cursor is on the error-causing block in the contour subprogram
■ Moving the zoom window	■ Repeat function not available	■ Repeat function available
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 FN17/FN18 or TABREAD-TABWRITE functions	■ Via FN17/FN18 or TABREAD-TABWRITE functions
■ Access to machine parameters	■ With the CFGREAD function	■ Via FN18 functions
■ Creating interactive cycles with CYCLE QUERY , e.g. touch-probe cycles in Manual Operation	■ Available	■ Not available

Tables and overviews

19.5 Functions of the TNC 640 and the iTNC 530 compared

Comparison: Differences in Test Run, functionality

Function	TNC 640	iTNC 530
Test Run up to block N	Function not available	Function available
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 block

Comparison: Differences in Test Run, operation

Function	TNC 640	iTNC 530
Arrangement of soft-key rows and soft keys within the rows	Arrangement of soft-key rows and soft-keys varies depending on the active screen layout.	
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
3-D view Displays a transparent workpiece	Available	Function not available
3-D view Displays a transparent tool	Available	Function not available
3-D view Displays tool paths	Available	Function not available
Adjustable model quality	Available	Function not available

Comparison: Differences in Manual Operation, functionality

Function	TNC 640	iTNC 530
Jog increment function	The jog increment can be defined separately for linear and rotary axes	The jog increment applies for both linear and rotary axes
Preset table	Basic transformation (translation and rotation) of machine table system to workpiece system via the columns X , Y and Z , as well as spatial angles SPA , SPB and SPC . In addition, the columns X_OFFS to W_OFFS can be used to define the axis offset of each individual axis. The function of the axis offsets can be configured.	Basic transformation (translation) of machine table system to workpiece system via the columns X , Y and Z , as well as a ROT basic rotation in the working plane (rotation). In addition, the columns A to W can be used to define datums in the rotary and parallel axes.

Functions of the TNC 640 and the iTNC 530 compared 19.5

Function	TNC 640	iTNC 530
Behavior during presetting	<p>Presetting in a rotary axis has the same effect as an axis offset. The offset is also effective for kinematics calculations and for tilting the working plane.</p> <p>The machine parameter CfgAxisPropKinn->presetToAlignAxis is used to define whether the axis offset is to be taken into account internally after zero setting.</p> <p>Independently of this, an axis offset has always the following effects:</p> <ul style="list-style-type: none"> ■ An axis offset always influences the nominal position display of the affected axis (the axis offset is subtracted from the current axis value). ■ If a rotary axis coordinate is programmed in an L block, then the axis offset is added to the programmed coordinate. 	<p>Rotary axis offsets defined by machine parameters do not influence the axis positions that were defined in the Tilt working plane function.</p> <p>MP7500 bit 3 defines whether the current rotary axis position referenced to the machine datum is taken into account, or whether a position of 0° is assumed for the first rotary axis (usually the C axis).</p>
Handling of preset table:		
<ul style="list-style-type: none"> ■ Editing the preset table in the Programming mode of operation ■ Preset tables that depend on the range of traverse 	<ul style="list-style-type: none"> ■ Not possible ■ Not available 	<ul style="list-style-type: none"> ■ Not possible ■ Available
Definition of feed-rate limitation	Feed-rate limitation can be defined separately for linear and rotary axes	Only one feed-rate limitation can be defined for linear and rotary axes

19.5 Functions of the TNC 640 and the iTNC 530 compared

Comparison: Differences in Manual Operation, operation

Function	TNC 640	iTNC 530
Capturing the position values from mechanical probes	Actual-position capture by soft key	Actual-position capture by hard key
Exiting the touch probe functions menu	Only via the END soft key	Via the END soft key or the END hard key

Comparison: Differences in Program Run, operation

Function	TNC 640	iTNC 530
Arrangement of soft-key rows and soft keys within the rows	Arrangement of soft-key rows and soft-keys varies depending on the active screen layout.	
Operating-mode switchover after program run was interrupted by switching to the Single Block mode of operation, and canceled by INTERNAL STOP	When you return to the Program Run mode of operation: Error message Selected block not addressed . Use mid-program startup to select the point of interruption	Switching the operating mode is allowed, modal information is saved, program run can be continued by pressing NC start
GOTO is used to go to FK sequences after program run was interrupted there before switching the operating mode	Error message FK programming: Undefined starting position	GOTO allowed
Entering with GOTO in the Program run single block	Function is only possible if the NC program was not yet started or after pressing the INTERNAL STOP soft key INTERNAL STOP	Function also possible after starting the NC program
Mid-program startup:		
<ul style="list-style-type: none"> ■ Behavior after restoring the machine status ■ Completing positioning for mid-program startup ■ Switching the screen layout for mid-program startup 	<ul style="list-style-type: none"> ■ The menu for returning must be selected with the RESTORE POSITION soft key ■ After position has been reached, positioning mode must be exited with the RESTORE POSITION soft key ■ Only possible, if startup position has already been approached 	<ul style="list-style-type: none"> ■ Menu for returning is selected automatically ■ The positioning mode is automatically exited after the position has been reached ■ Possible in all operating states
Error messages	Error messages are still active after the error has been corrected and must be acknowledged separately	Error messages are sometimes acknowledged automatically after the error has been corrected
Point patterns in single block	With point pattern cycles and CYCL CALL PAT , the control stops after each point.	Point pattern cycles and CYCL CALL PAT are handled by the control as a single block

Comparison: Differences in Program Run, traverse movements



Caution: Check the traverse movements!

NC programs that were created on earlier TNC controls may lead to different traverse movements or error messages on a TNC 640!

Be sure to take the necessary care and caution when running-in programs!

Please find a list of known differences below. The list does not pretend to be complete!

Function	TNC 640	iTNC 530
Handwheel-superimposed traverse with M118	Effective in the active coordinate system (which may also be rotated or tilted), or in the machine-based coordinate system, depending on the setting in the 3-D ROT menu for manual operation	Effective in the machine-based coordinate system
Approach/Departure with APPR/DEP , RO is active, contour element plane is not equal to working plane	If possible, the blocks are executed in the defined contour element plane , error message for APPRLN , DEPLN , APPRCT , DEPCT	If possible, the blocks are executed in the defined working plane ; error message for APPRLN , APPRLT , APPRCT , APPRLCT
Scaling approach/departure movements (APPR/DEP/RND)	Axis-specific scaling factor is allowed, radius is not scaled	Error message
Approach/departure with APPR/DEP	Error message if RO is programmed for APPR/DEP LN or APPR/DEP CT	Tool radius 0 and compensation direction RR are assumed
Approach/departure with APPR/DEP if contour elements with length 0 are defined	Contour elements with length 0 are ignored. The approach/departure movements are calculated for the first or last valid contour element	An error message is issued if a contour element with length 0 is programmed after the APPR block (relative to the first contour point programmed in the APPR block) For a contour element with length 0 before a DEP block, the TNC does not issue an error message, but uses the last valid contour element to calculate the departure movement

19.5 Functions of the TNC 640 and the iTNC 530 compared

Function	TNC 640	iTNC 530
Effect of Q parameters	Q60 to Q99 (or QS60 to QS99) are always local	Q60 to Q99 (or QS60 to QS99) are local or global, depending on MP7251 in converted cycle programs (.cyc). Nested calls may cause problems
Automatic cancelation of tool radius compensation	<ul style="list-style-type: none"> ■ Block with R0 ■ DEP block ■ END PGM 	<ul style="list-style-type: none"> ■ Block with R0 ■ DEP block ■ PGM CALL ■ Programming of Cycle 10 ROTATION ■ Program selection
NC blocks with M91	No consideration of tool radius compensation	Consideration of tool radius compensation
Tool shape compensation	Tool shape compensation is not supported, because this type of programming is considered to be axis-value programming, and the basic assumption is that axes do not form a Cartesian coordinate system	Tool shape compensation is supported
Mid-program startup in a point table	The tool is positioned above the next position to be machined	The tool is positioned above the last position that has been completely machined
Empty CC block (pole of last tool position is used) in NC program	Last positioning block in the working plane must contain both coordinates of the working plane	Last positioning block in the working plane does not necessarily need to contain both coordinates of the working plane. Can cause problems with RND or CHF blocks
Axis-specific scaling of RND block	RND block is scaled, the result is an ellipse	Error message is issued
Reaction if a contour element with length 0 is defined before or after a RND or CHF block	Error message is issued	<p>Error message is issued if a contour element with length 0 is located before the RND or CHF block</p> <p>Contour element with length 0 is ignored if the contour element with length 0 is located after the RND or CHF block</p>

Functions of the TNC 640 and the iTNC 530 compared 19.5

Function	TNC 640	iTNC 530
Circle programming with polar coordinates	The incremental rotation angle IPA and the direction of rotation DR must have the same sign. Otherwise, an error message will be issued	The algebraic sign of the direction of rotation is used if the sign defined for DR differs from the one defined for IPA
Tool radius compensation on circular arc or helix with angular length = 0	The transition between the adjacent elements of the arc/helix is generated. Also, the tool axis motion is executed right before this transition. If the element is the first or last element to be corrected, the next or previous element is dealt with in the same way as the first or last element to be corrected	The equidistant line of the arc/helix is used for generating the tool path
Compensation of tool length in the position display	The values L and DL from the tool table and the value DL from the TOOL CALL are taken into account in the position display	The values L and DL from the tool table are taken into account in the position display
Traverse movement in spacial arc	Error message is issued	No restrictions
SLII Cycles 20 to 24:		
■ Number of definable contour elements	■ Max. 16384 blocks in up to 12 subcontours	■ Max. 8192 contour elements in up to 12 subcontours, no restrictions for subcontour
■ Define the working plane	■ Tool axis in TOOL CALL block defines the working plane	■ The axes of the first positioning block in the first subcontour define the working plane
■ Position at end of SL cycle	■ End position = clearance height above the last position that is defined before the cycle call	■ With MP7420, you can define whether the end position is above the last programmed position, or whether the tool moves only to clearance height

Tables and overviews

19.5 Functions of the TNC 640 and the iTNC 530 compared

Function	TNC 640	iTNC 530
SLII Cycles 20 to 24:		
<ul style="list-style-type: none"> ■ Handling of islands which are not contained in pockets ■ Set operations for SL cycles with complex contour formulas ■ Radius compensation is active during CYCL CALL ■ Paraxial positioning blocks in contour subprogram ■ Miscellaneous functions M in contour subprogram ■ M110 (feed-rate reduction for inside corner) 	<ul style="list-style-type: none"> ■ Cannot be defined with complex contour formula ■ Real set operation possible ■ Error message is issued ■ Error message is issued ■ Error message is issued ■ Function does not work within SL cycles 	<ul style="list-style-type: none"> ■ Restricted definition in complex contour formula is possible ■ Only restricted performance of real set operation possible ■ Radius compensation is canceled, program is executed ■ Program is executed ■ M functions are ignored ■ Function also works within SL cycles
General cylinder surface machining:		
<ul style="list-style-type: none"> ■ Contour definition ■ Offset definition on cylinder surface ■ Offset definition for basic rotation ■ Circle programming with C/CC ■ APPR/DEP blocks in contour definition 	<ul style="list-style-type: none"> ■ With X/Y coordinates, independent of machine type ■ With datum shift in X/Y, independent of machine type ■ Function available ■ Function available ■ Function not available 	<ul style="list-style-type: none"> ■ Machine-dependent, with existing rotary axes ■ Machine-dependent datum shift in rotary axes ■ Function not available ■ Function not available ■ Function available
Cylinder surface machining with Cycle 28:		
<ul style="list-style-type: none"> ■ Complete roughing-out of slot ■ Definable tolerance 	<ul style="list-style-type: none"> ■ Function available ■ Function available 	<ul style="list-style-type: none"> ■ Function not available ■ Function available
Cylinder surface machining with Cycle 29	Direct plunging to contour of ridge	Circular approach to contour of ridge
Cycles 25x for pockets, studs and slots:		
<ul style="list-style-type: none"> ■ Plunging movements 	In limit ranges (geometrical conditions of tool/contour) error messages are triggered if plunging movements lead to unreasonable/critical behavior	In limit ranges (geometrical conditions of tool/contour), vertical plunging is used if required

Functions of the TNC 640 and the iTNC 530 compared 19.5

Function	TNC 640	iTNC 530
PLANE function:		
<ul style="list-style-type: none"> ■ TABLE ROT/COORD ROT not defined ■ Machine is configured for axis angle ■ Programming an incremental spatial angle according to PLANE AXIAL ■ Programming an incremental axis angle according to PLANE SPATIAL if the machine is configured for spatial angle ■ Programming of PLANE functions with active Cycle 8 MIRROR IMAGE 	<ul style="list-style-type: none"> ■ Configured setting is used ■ All PLANE functions can be used ■ Error message is issued ■ Error message is issued ■ Error message is issued ■ PLANE AXIAL is possible 	<ul style="list-style-type: none"> ■ COORD ROT is used ■ Only PLANE AXIAL is executed ■ Incremental spatial angle is interpreted as an absolute value ■ Incremental axis angle is interpreted as an absolute value ■ Function is available with all PLANE functions
Special functions for cycle programming:		
<ul style="list-style-type: none"> ■ FN17 ■ FN18 	<ul style="list-style-type: none"> ■ Function available, details are different ■ Function available, details are different 	<ul style="list-style-type: none"> ■ Function available, details are different ■ Function available, details are different
Compensation of tool length in the position display	The tool length entries L and DL from the tool table are taken into account in the position display, from TOOL CALL depending on the machine parameter progToolCallIDL	The tool length entries L and DL from the tool table are taken into account in the position display

Comparison: Differences in MDI operation

Function	TNC 640	iTNC 530
Execution of connected sequences	Function partially available	Function available
Saving modally effective functions	Function partially available	Function available

19.5 Functions of the TNC 640 and the iTNC 530 compared

Comparison: Differences in programming station

Function	TNC 640	iTNC 530
Demo version	Programs with more than 100 NC blocks cannot be selected, an error message is issued	Programs can be selected, max. 100 NC blocks are displayed, further blocks are truncated in the display
Demo version	If nesting with PGM CALL results in more than 100 NC blocks, there is no test graphic display; an error message is not issued	Nested programs can be simulated.
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 the soft-key bar shifts the soft-key row to the right, or to the left	Clicking any soft-key bar activates the respective soft-key row

Index

3

3-D basic rotation.....	561
3D compensation.....	474
Delta values.....	476
Face Milling.....	477
Normalized vector.....	475
Peripheral Milling.....	478
Tool orientation.....	476
3-D compensation	
Tool shapes.....	476
3-D touch probes	
Calibration.....	553
3-D view.....	587

A

About this manual.....	6
ACC.....	412
Accessing tables.....	331
Accessories.....	93
Actual position capture.....	108
Adaptive feed control.....	400
Adding comments.....	142, 144
Additional axes.....	97, 97
Adjusting spindle speed.....	531
AFC.....	400
Align tool axis.....	458
Angle functions.....	305
Approach contour.....	212
ASCII Files.....	423
Automatic program start.....	608
Automatic tool measurement...	177

B

Basic rotation.....	560
Measuring in Manual Operation	
mode.....	560
Behavior after reception of ETX	626
Block.....	110
Delete.....	110
Block Check Character.....	626

C

CAD Viewer.....	259
CAD viewer and DXF converter	
screen layout.....	258
Calculating with parentheses...	341
Calculation of circles.....	306
Calculator.....	146
CAM programming.....	474
Chamfer.....	224
Chatter Control.....	412
Checking the axis positions.....	534
Circle.....	228, 230, 236
Circle center.....	226
Circular path.....	227, 236
Code numbers.....	623

Collision monitoring.....	393
Comparison.....	673
Compensating workpiece	
misalignment	
By measuring two points on a	
straight surface.....	559
Condition of RTS line.....	626
Connecting/removing USB devices.	139
Connector pin layout for data	
interfaces.....	658
Context-sensitive help.....	162
Control panel.....	74
Conversational dialog.....	106
Coordinate transformation.....	420
Copying program sections 111, 111	
Corner rounding.....	225

D

Data Backup.....	116
Data interface.....	624
Connector pin layouts.....	658
Set up.....	624
Data output on the screen.....	319
Data transfer software.....	629
Data transfer speed....	
624, 625, 625, 625, 625, 626, 626	
Datum management.....	536
Datum setting.....	543
Without a 3-D touch probe....	543
Datum shift	
Resetting.....	422
Using the datum table.....	421
Datum table.....	551
Transferring test results.....	551
DCM.....	393
Defining local Q parameters....	301
Defining nonvolatile Q parameters..	301
Defining the workpiece blank...	104
Depart contour.....	212
Dialog.....	106
Directory.....	117, 121
Copy.....	123
Create.....	121
Delete.....	125
Displaying HTML files.....	132
Displaying Internet files.....	132
Display screen.....	73
Downloading help files.....	167
Dwell time.....	433, 434
DXF converter.....	260
Selecting hole positions	
Icon.....	275
Mouse area.....	274
Dynamic Collision Monitoring...	393

E

Enter spindle speed.....	186
Error messages.....	156, 156
Help with.....	156
Ethernet interface.....	631
Configuring.....	631
Connecting and disconnecting	
network drives.....	138
Connection options.....	631
Introduction.....	631
External access.....	615
External data transfer	
iTNC 530.....	136

F

FCL.....	623
FCL function.....	11
Feature Content Level.....	11
Feed control, automatic.....	400
Feed rate.....	530
Adjust.....	531
Input options.....	107
On rotary axes, M116.....	461
Feed rate factor for plunging	
movements M103.....	375
Feed rate in millimeters per spindle	
revolution M136.....	376
File	
Create.....	121
File functions.....	419
File Management.....	114, 117
File Manager	
Calling.....	119
File manager	
Copying files.....	121
Copying tables.....	123
Delete file.....	125
Directories.....	117
Copy.....	123
Create.....	121
External data transfer.....	136
File	
Create.....	121
File type.....	114
File type	
External file types.....	116
Function overview.....	118
Overwriting files.....	122
Protect file.....	128
Rename file.....	127, 127
Selecting files.....	120
Tagging files.....	126
File status.....	119
Filter for hole positions with DXF	
data update.....	276
Firewall.....	
FK programming.....	241, 241
Circular paths.....	246

Fundamentals.....	241	Helical interpolation.....	237	Nesting.....	289
FK-Programming		Helix.....	237	Network connection.....	138
Graphics.....	243	Help system.....	162	Network settings.....	631
FK programming		Help with error messages.....	156	O	
Initiating dialog.....	244	I		Open BMP file.....	135
Input options.....	247	Inclined-tool machining in a tilted		Open contour corners M98.....	374
Auxiliary points.....	250	plane.....	459	Open GIF file.....	135
Circle data.....	248	Inclined turning.....	512	Opening a video file.....	134
Closed contours.....	249	Initiated tools.....	180	Opening Excel files.....	131
Direction and length of		Inserting and modifying blocks. 110		Opening graphic files.....	135
contour elements.....	247	Interrupt machining.....	600	Opening TXT files.....	134
End points.....	247	iTNC 530.....	72	Open INI file.....	134
Relative data.....	251	L		Open JPG file.....	135
Straight lines.....	245	Load machine configuration.....	643	Open PNG file.....	135
FN14: ERROR: Displaying error		Look ahead.....	378	Open TXT file.....	134
messages.....	312, 312	M		Operating times.....	622
FN16: F-PRINT: Output of formatted		M91, M92.....	370	Option number.....	623
texts.....	316, 316	Machine settings.....	615	P	
FN18: SYSREAD: Reading system		Manual datum setting.....	563	Pallet table.....	482
data.....	320, 320	Circle center as datum.....	565	Application.....	482
FN19: PLC: Transfer values to the		Corner as datum.....	564	Run.....	484
PLC.....	329, 329	In any axis.....	563	Select and exit.....	484
FN20: WAIT FOR: NC and PLC		Setting a center line as datum	567	Transfer coordinates.....	482, 482
synchronization.....	329	Measurement of machining		Parallel axes.....	414
FN23: CIRCLE DATA: Calculate a		time.....	593	Parameter programming:See Q	
circle from 3 points.....	306	Measuring workpieces.....	568	parameter programming... 298, 345	
FN24: CIRCLE DATA: Calculate a		M functions		Paraxcomp.....	414
circle from 4 points.....	306	For program run inspection....	369	Paraxmode.....	414
FN26: TABOPEN: Open a freely		For spindle and coolant.....	369	Part families.....	302
definable table.....	430	See miscellaneous functions..	368	Path.....	117
FN27: TABWRITE: Write to a freely		Mid-program startup.....	605	Path contours.....	222
definable table.....	431, 431	After power failure.....	605	Cartesian coordinates.....	222
FN28: TABREAD: Read from a		Miscellaneous functions.....	368	Circle with tangential	
freely definable table.....	432, 432	enter.....	368	connection.....	230
FN29: PLC: Transfer values to the		For coordinate data.....	370	Circular path around circle	
PLC.....	330	For path behavior.....	373	center CC.....	227
FN37: EXPORT.....	330	For rotary axes.....	461	Circular path with defined	
Formatted output of Q parameter		Modes of Operation.....	75	radius.....	228
values.....	316	MOD function.....	612	Overview.....	222
Form view.....	429	Exit.....	612	Straight line.....	223
Freely definable tables.....		Overview.....	613	Polar coordinates.....	234
FS, Functional safety.....	532	Select.....	612	Circular path around pole	
Full circle.....	227	Monitoring		CC.....	236
Functional safety FS.....	532	Collision.....	393	Circular path with tangential	
Fundamentals.....	96	Move machine axes		connection.....	236
G		Jog positioning.....	519	Overview.....	234
Graphics.....	584	Moving the axes		Straight line.....	235
Display modes.....	586	With machine axis direction		Path functions.....	206
With programming.....	152	buttons.....	519	Fundamentals.....	206
Magnification of details....	155	Moving the machine axes.....	519	Circles and circular arcs....	209
Graphic settings.....	614	with the handwheel.....	520	Pre-position.....	210
Graphic simulation.....	592	Multiple axis machining.....	469	PDF Viewer.....	129
Tool display.....	592	N		PLANE Function.....	437, 438
H		NC and PLC synchronization....	329	PLANE function	
Handwheel.....	520	NC error messages.....	156	Automatic positioning.....	453
Hard disk.....	114			Axis angle definition.....	451

Euler angle definition.....	444
Inclined-tool machining.....	459
Incremental definition.....	450
Point definition.....	448
Positioning behavior.....	453
Projection angle definition.....	443
Reset.....	440
Selection of possible solutions....	456
Spatial angle definition.....	441
Vector definition.....	446
Plan view.....	590
PLC and NC synchronization....	329
Pocket table.....	183
Polar coordinates.....	98
Fundamentals.....	98
Programming.....	234
Positioning.....	578
With Manual Data Input.....	578
With tilted working plane....	372, 468
Preset table.....	536, 552
Transferring test results.....	552
Principal axes.....	97, 97
Probing a plane.....	561
Processing DXF data	
Basic settings.....	263
Filter for hole positions.....	276
Selecting a contour.....	268
Selecting hole positions	
Single selection.....	273
Selecting machining positions	272
Setting layers.....	265
Setting the datum.....	266
Program.....	101
Editing.....	109
Opening a new program.....	104
Organization.....	101
Structuring.....	145
Program call	
Any desired program as	
subprogram.....	285
Program defaults.....	391
Program Management:See File	
Management.....	114
Programming graphics.....	243
Programming tool movements.	106
Program run.....	598
Execute.....	599
Interrupt.....	600
Mid-program startup.....	605
Optional block skip.....	609
Overview.....	598
Resuming after interruption....	601
Retraction.....	603
Program-section repeat.....	283
Projection in three planes.....	590
Protection zone.....	617

Q

Q parameter	
Export.....	330
Transfer values to the	
PLC.....	329, 330
Q parameter programming....	298, 345
Additional functions.....	311
Angle functions.....	305
Calculation of circles.....	306
If-then decisions.....	307
Mathematical functions.....	303
Programming notes....	
300, 346, 347, 348, 350, 352	
Q parameters.....	298, 345
Checking.....	309
Local parameters QL.....	298
Nonvolatile parameters QR....	298
Preassigned.....	356

R

Radius compensation.....	201
Entering.....	202
Outside corners, inside	
corners.....	203
Rapid traverse.....	170
Reading out machine parameters....	353
Recessing and undercutting....	505
Reference system.....	97, 97
Replacing texts.....	113
Retraction.....	603
After a power interruption.....	603
Retraction from the contour.....	382
Returning to the contour.....	607
Rotary axis.....	461
Reduce display M94.....	463
Shortest-path traverse: M126.	462
Rounding corners M197.....	387

S

Screen layout.....	73
Search function.....	112
Selecting a contour from DXF..	268
Selecting positions from DXF..	272
Selecting the datum.....	100
Selecting the unit of measure..	104
Select kinematics.....	619
Setting the BAUD RATE....	
624, 625, 625, 625, 625,	
626, 626, 626, 626, 626	
Software number.....	623
SPEC FCT.....	390
Special functions.....	390
Spindle load monitoring.....	411
SQL commands.....	331
Status display.....	78
Additional.....	79

General.....	78
Straight line.....	223, 235
String parameters.....	345
Structuring programs.....	145
Subprogram.....	281
Superimposing handwheel	
positioning M118.....	380
Surface normal vector....	
446, 460, 474,	475
Switch-off.....	518
Switch-on.....	516

T

TCPM.....	469
Reset.....	473
Teach In.....	108, 223
Teach-in cut.....	404
Test Run.....	595
Test run	
Execute.....	597
Test Run	
Overview.....	595
test run	
Setting speed.....	585
Text File.....	423
Text file	
Delete functions.....	424
Finding text sections.....	426
Opening and exiting.....	423
Text variables.....	345
Tilted axes.....	464
Tilting the Working Plane....	
437, 438,	571
Tilting the working plane	
Manual.....	571
Tilting without rotary axes.....	458
TNCguide.....	162
TNCremo.....	629
TNCremoNT.....	629
Tool breakage monitoring.....	410
Tool change.....	188
Tool compensation.....	200
Length.....	200
Tool Compensation	
Radius.....	201
Tool compensation	
Three-dimensional.....	474
Tool data.....	172
Call.....	186
Delta values.....	173
Entering into the program.....	173
Enter into the table.....	174
Tool data	
Initiating.....	180
Tool length.....	172
Tool management.....	192
Tool measurement.....	177
Tool name.....	172

Index

Tool number.....	172
Tool radius.....	172
Tool table.....	174
Edit, exit.....	178
Editing functions.....	180, 195, 196
Input options.....	174
Tool usage file.....	190, 618
Tool usage test.....	190
Touch probe cycles.....	545
Manual Operation mode.....	545
See Touch Probe Cycles User's Manual	
Touch probe monitoring.....	384
TRANS DATUM.....	420
Traverse limits.....	617
Traversing reference marks.....	516
Trigonometry.....	305
Turning	
Tool data.....	499
Turning mode selection.....	489
Turning Operations.....	488
Feed rate.....	493
Turning operations	
Program spindle speed.....	492
Turning Operations	
Tool tip radius compensation..	504
T vector.....	475

U

Unbalance functions.....	494
User parameters	
Machine-specific.....	646
Using touch probe functions with mechanical probes or measuring dials.....	544

V

Version numbers.....	623, 643
Virtual tool axis.....	381

W

Window Manager.....	86
Wireless handwheel.....	523
Assign handwheel holder.....	640
Configure.....	640
Selecting transmitter power...	641
Setting channel.....	641
Statistical data.....	642
Working space monitoring	594, 597
Workpiece positions.....	99
Writing probing values in a datum table.....	551
Writing probing values in a preset table.....	552

Z

Zero point shift.....	420
Coordinate input.....	420
ZIP archive.....	133

HEIDENHAIN

DR. JOHANNES HEIDENHAIN GmbH

Dr.-Johannes-Heidenhain-Straße 5

83301 Traunreut, Germany

☎ +49 8669 31-0

FAX +49 8669 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

TNC 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

Lathe controls ☎ +49 8669 31-3105

E-mail: service.lathe-support@heidenhain.de

www.heidenhain.de

Touch probes from HEIDENHAIN

help you reduce non-productive time and
improve the dimensional accuracy of the finished workpieces.

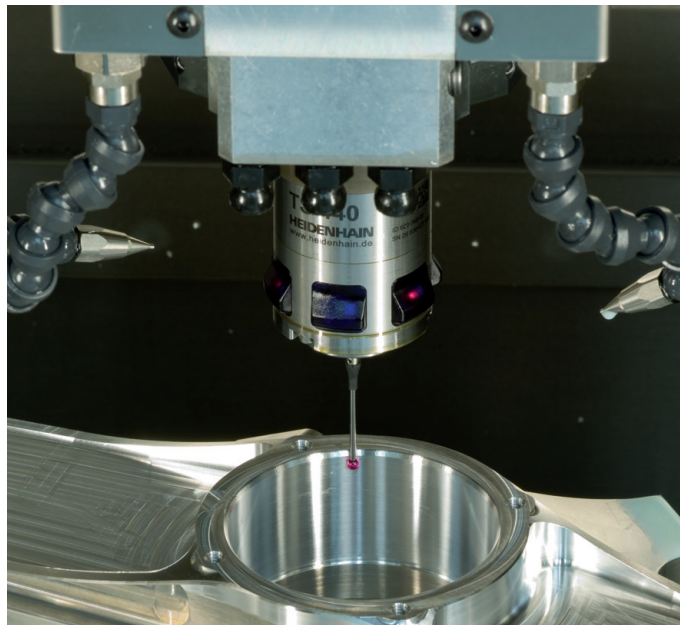
Workpiece touch probes

TS 220 Signal transmission by cable

TS 440, TS 444 Infrared transmission

TS 640, TS 740 Infrared transmission

- Workpiece alignment
- Setting datums
- Workpiece measurement



Tool touch probes

TT 140 Signal transmission by cable

TT 449 Infrared transmission

TL Contact-free laser systems

- Tool measurement
- Wear monitoring
- Tool breakage detection

